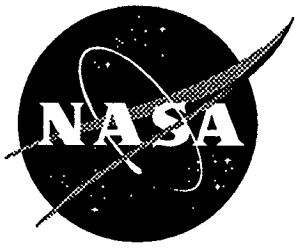
NASA Technical Memorandum 110332



ZIP2DL: An Elastic-Plastic, Large-Rotation Finite-Element Stress Analysis and Crack-Growth Simulation Program

Xiaomin Deng
University of South Carolina, Columbia, South Carolina

James C. Newman, Jr.
Langley Research Center, Hampton, Virginia

April 1997

National Aeronautics and
Space Administration
Langley Research Center
Hampton, Virginia 23681-0001

TABLE OF CONTENTS

ZIP2DL:
An Elastic-Plastic, Large-Rotation Finite-Element Stress Analysis and Crack-Growth Simulation Program

1. SUMMARY

ZIP2DL is a two-dimensional, elastic-plastic finite element program for stress analysis and crack growth simulations, developed for the NASA Langley Research Center. It has many of the salient features of the ZIP2D program [1, 2]. For example, ZIP2DL contains five material models (linearly elastic, elastic-perfectly plastic, power-law hardening, linear hardening, and multi-linear hardening models), and it can simulate mixed-mode crack growth for prescribed crack growth paths under plane stress, plane strain, and mixed state of stress conditions. Further, as an extension of ZIP2D, it also includes a number of new capabilities. The large-deformation kinematics in ZIP2DL will allow it to handle elastic problems with large strains and large rotations, and elastic-plastic problems with small strains and large rotations. Loading conditions in terms of surface traction, concentrated load, and nodal displacement can be applied with a default linear time dependence or they can be programmed according to a user-defined time dependence through a user subroutine. The restart capability of ZIP2DL will make it possible to stop the execution of the program at any time, analyze the results and/or modify execution options, and resume and continue the execution of the program.

This report includes three sections: a theoretical manual section, a user manual section, and an example manual section. In the theoretical manual section, the mathematics behind the various aspects of the program are concisely outlined. In the user manual section, a line-by-line explanation of the input data is given. In the example manual section, three types of examples are presented to demonstrate the accuracy and illustrate the usage of this program.

2. THEORETICAL MANUAL

2.1 INTRODUCTION

ZIP2DL has the following capabilities:

- Stress analysis and mixed-mode stable crack growth simulation.
- Plane stress, plane strain, and mixed state of stress (plane stress with a plane strain core, useful in crack growth simulations).
- Five common material models (linearly elastic, elastic-perfectly plastic, power-law hardening, linear hardening or bi-linear, and multi-linear hardening models).
- Small-deformation (small strain and small rotation), linear, elastic analysis; large-deformation (large strain and/or large rotation), linear, elastic analysis; small-deformation elastic-plastic analysis; and small-strain, large-rotation, elastic-plastic analysis.
- Reduced integration for mesh-locking prevention.
- Surface traction loading, nodal force loading, and nodal displacement loading.
- User subroutine for loading time-dependence definition.
- Restart file capability.

ZIP2DL is formulated with the principle of virtual work. Large deformation kinematics is described by a total Lagrangian formulation. J_2 flow theory of plasticity is employed. Integration of the incremental stress-strain relationship is performed using a tangent predictor-radial return algorithm with adaptive subincrementation. An effective measure that prevents the occurrence of negative plastic flow and enhance the overall numerical accuracy and convergence is implemented. A reduced integration option is included for plane strain problems with dominant incompressible deformation. Three types of elements are available (only one type can be used for a particular problem): three-noded constant-strain triangular elements, and four- and eight-noded isoparametric quadrilateral elements. Crack growth is simulated with a nodal release procedure which is activated when a critical crack-tip-opening displacement (CTOD) fracture criterion is satisfied. Finite element equations are cast in forms consistent with either the "initial-stress"

approach (for small-deformation and small-strain, large-rotation elastic-plastic problems) or the "tangent-stiffness" approach (for small-deformation and small-strain, large-rotation elastic-plastic problems, and for large-deformation elastic problems). In an "initial-stress" approach, as discussed later in detail, nonlinear terms in finite element equations due to plasticity are treated as a contribution to the force vector, while nonlinear terms due to large-deformation are treated as a contribution to the stiffness matrix. In a "tangent-stiffness" approach, on the other hand, all nonlinear terms are treated as contributions to the stiffness matrix. In either case, the finite element equations are solved with the Newton-Raphson method or its modified versions.

2.2 FINITE ELEMENT FORMULATIONS

This section derives two integral representations of the boundary-value problem in question based on the principle of virtual work. These two representations lead to two equivalent forms of the finite element equations and are the basis for the two solution approaches to be discussed in detail in Section 2.3.

Consider a body undergoing an arbitrary deformation. Suppose that at the current time t the body occupies a domain Ω with boundary Γ (the *current configuration*) and that at an initial time t_0 it occupies a domain Ω_0 with boundary Γ_0 (the *reference configuration*). Furthermore, suppose that a material particle is described by its position $\boldsymbol{X}$ at t_0 and position $\boldsymbol{x}=\boldsymbol{x}(\boldsymbol{X}, t)$ at t, and the displacement of the particle at t relative to its position at t_0 is $\boldsymbol{u}=\boldsymbol{u}(\boldsymbol{X}, t)$, so that $\boldsymbol{x}(\boldsymbol{X}, t)=\boldsymbol{X}+\boldsymbol{u}(\boldsymbol{X}, t)$. Following conventions (e.g. [3, 4]), we now define our stress and strain measures. The *deformation gradient tensor* $\boldsymbol{F}$ is defined by

$$F = x\bar{\nabla} = I + u\bar{\nabla}, \quad \text{or} \quad F_{iJ} = \frac{\partial x_i}{\partial X_J} = \delta_{iJ} + \frac{\partial u_i}{\partial X_J} \tag{1}$$

where $\mathbf{I}$ is the second-order identity tensor. The *Green-Lagrange strain tensor* $\boldsymbol{E}$ is given by

$$E = \frac{1}{2}\left(F^T \cdot F - I\right) = \frac{1}{2}\left[u\bar{\nabla} + \bar{\nabla}u + \left(\bar{\nabla}u\right)\cdot\left(u\bar{\nabla}\right)\right] \tag{2}$$

where superscript "T" denotes transposition. Let $\boldsymbol{\sigma}$ be the (symmetric) *Cauchy stress tensor*, then the symmetric *second Piola-Kirchhoff stress tensor*, $\boldsymbol{S}$, can be defined by

$$S = JF^{-1} \cdot \sigma \cdot \left(F^{-1}\right)^T \tag{3}$$

where J is the *Jacobian* and is given by J=det$\boldsymbol{F}$, which is the determinant of $\boldsymbol{F}$.

The *Principle of Virtual Work* at time t+Δt is given by

$$\int_{\Omega_0} S(t+\Delta t) : \delta E d\Omega_0 = \delta W(t+\Delta t) \tag{4}$$

where on the left-hand side is the *internal virtual work* stated in the reference configuration; δW is the *external virtual work* done by external forces (e.g. boundary surface tractions) on $\delta \boldsymbol{u}$; and $\delta \boldsymbol{u}$ and $\delta \boldsymbol{E}$ are, respectively, the *virtual displacement vector* and the *associated virtual strain tensor*. It is worth noting that $\delta \boldsymbol{u}$ and $\delta \boldsymbol{E}$ are not related to each other by simply replacing $\boldsymbol{u}$ and $\boldsymbol{E}$ in Eq. (2) with $\delta \boldsymbol{u}$ and $\delta \boldsymbol{E}$, respectively. Their relationship will be given later. To develop an iterative finite element algorithm we express the value of any quantity at time t+Δt as the sum of its value at time t and its increment at time t, so that we can write, for the stress tensor, displacement vector and strain tensor in the preceding equations,

$$S(t+\Delta t) = S + \Delta S,\ u(t+\Delta t) = u + \Delta u,\ E(t+\Delta t) = E + \Delta E \tag{5}$$

where the strain tensor increment ΔE is related to the displacement vector increment Δu as follows:

$$\begin{aligned}
&\Delta E = \Delta\varepsilon + \Delta\varepsilon^{*} + \Delta\varepsilon^{**},\\
&\Delta\varepsilon = \frac{1}{2}\left[(\Delta u)\vec{\nabla} + \vec{\nabla}(\Delta u)\right],\\
&\Delta\varepsilon^{*} = \frac{1}{2}\left[\vec{\nabla}u \cdot (\Delta u)\vec{\nabla} + \vec{\nabla}(\Delta u) \cdot u\vec{\nabla}\right],\\
&\Delta\varepsilon^{**} = \frac{1}{2}\vec{\nabla}(\Delta u) \cdot (\Delta u)\vec{\nabla}
\end{aligned} \tag{6}$$

Similarly, the virtual displacement vector and virtual strain tensor can be written as

$$\delta u = \delta\Delta u,\ \delta E = \delta\varepsilon + \delta\varepsilon^{*} + \delta\varepsilon^{**},$$

$$\delta\varepsilon = \frac{1}{2}\left[(\delta\Delta u)\vec{\nabla} + \vec{\nabla}(\delta\Delta u)\right],$$

$$\delta\varepsilon^{*} = \frac{1}{2}\left[\vec{\nabla}u\cdot(\delta\Delta u)\vec{\nabla} + \vec{\nabla}(\delta\Delta u)\cdot u\vec{\nabla}\right], \tag{7}$$

$$\delta\varepsilon^{**} = \frac{1}{2}\left[\vec{\nabla}(\Delta u)\cdot(\delta\Delta u)\vec{\nabla} + \vec{\nabla}(\delta\Delta u)\cdot(\Delta u)\vec{\nabla}\right]$$

Substitution of Eqs. (5)-(7) into Eq. (4) leads to

$$\int_{\Omega_0}\left[\Delta S:\delta E + S:\delta\varepsilon^{**}\right]d\Omega_0 = \delta W - \int_{\Omega_0} S:\left(\delta\varepsilon + \delta\varepsilon^{*}\right)d\Omega_0 \tag{8}$$

A generalization of the Hooke's law to arbitrarily large deformation cases is given by

$$S = C:E \quad \text{or} \quad \Delta S = C:\Delta E \tag{9}$$

where $\boldsymbol{C}$ is the conventional fourth-order elasticity tensor. To apply the theory of plasticity, we note that, when measured in the reference configuration Ω_0, the tensors $\boldsymbol{E}$ and $\boldsymbol{S}$ are invariant under arbitrary rigid-body rotations. As such, they can be used to replace engineering strains and stresses in small-strain constitutive equations under small-strain, large-rotation conditions. For example, a generalization of an incremental theory of plasticity (e.g. the J_2 flow theory of plasticity used in the current study) to small-strain, large-rotation cases can be written as

$$\Delta S = C:\left(\Delta E - \Delta E^{P}\right) \quad \text{or} \quad \Delta S = C^{el-pl}:\Delta E \tag{10}$$

where $\Delta\boldsymbol{E}^{P}$ is the plastic strain increment tensor and $\boldsymbol{C}^{el\text{-}pl}$ is the fourth-order elastic-plastic tangent modulus tensor (see Section 2.4). An approach for elastic-plastic computation under small-strain and large-rotation conditions is to substitute the first equation in (10), move the plasticity induced nonlinear term to the right-hand side of the equation as an equivalent body force, and drop all higher-order nonlinear terms related to $\Delta\varepsilon^{**}$ and $\delta\varepsilon^{**}$, so that

$$\int_{\Omega_0}\left(\Delta\varepsilon + \Delta\varepsilon^{*}\right):C:\left(\delta\varepsilon + \delta\varepsilon^{*}\right)d\Omega_0 = \delta W - \int_{\Omega_0} S:\left(\delta\varepsilon + \delta\varepsilon^{*}\right)d\Omega_0 + \int_{\Omega_0}\Delta E^{P}:C:\left(\delta\varepsilon + \delta\varepsilon^{*}\right)d\Omega_0 \tag{11}$$

Equation (11) can be viewed as an extension of the "initial stress" method [5] to small-strain and large-rotation deformation problems and can be properly termed an *initial-stress based total Lagrangian finite element formulation*. The resulting nonlinear finite element matrix equation is as follows:

$$[K]\{\Delta U\} = \{\Delta f\}, \text{ where } [K] = [K^1] + [K^2],\ \{\Delta f\} = \{\Delta f^1\} + \{\Delta f^2\} \tag{12}$$

where $\{\Delta U\}$ is the vector of unknown nodal displacement increments; $[K^1]$ is the constant stiffness matrix for a linearly elastic, small-deformation (small-strain and small-rotation) problem; $[K^2]$ is the contribution to the stiffness matrix from nonlinear terms due to large-deformation (large-strain and/or large rotation) kinematics, which depends on the current state of stress and deformation; $\{\Delta f\}$ is the total nodal force vector, in which $\{\Delta f^1\}$ is a vector of unbalanced nodal forces due to external loads and interal stresses and is zero at equilibrium; and $\{\Delta f^2\}$ is a vector of nodal forces due to plastic strain increments and is zero for a converged solution. Alternatively, an approach based on the "tangent-stiffness" method [6] is to use the second equation in (10) and keep the plastic nonlinear term on the left-hand side of the equation to obtain

$$\int_{\Omega_0} \left(\Delta\varepsilon + \Delta\varepsilon^*\right) : C^{el-pl} : \left(\delta\varepsilon + \delta\varepsilon^*\right) d\Omega_0 = \delta W - \int_{\Omega_0} S : \left(\delta\varepsilon + \delta\varepsilon^*\right) d\Omega_0 \tag{13}$$

This is a *classical tangent-stiffness based total Lagrangian finite element formulation.* The resulting nonlinear finite element matrix equation is given by

$$[K]\{\Delta U\} = \{\Delta f\}, \text{ where } [K] = [K^1] + [K^2] + [K^3],\ \{\Delta f\} = \{\Delta f^1\} \tag{14}$$

where $[K^3]$, the additional term in the total stiffness matrix $[K]$, is the contribution to the stiffness matrix due to plasticity and depends on the current stress state. All other quantities have the same meaning as those in Eq. (12). It is noted that the effect of plasticity is treated as a stiffness contribution in Eq. (14) instead of a force contribution as in Eq. (12).

Element level expressions for the stiffness matrices and force vectors in Eqs. (12) and (14) can be obtained from Eqs. (11) and (13), respectively, using standard finite element discretization procedures. In Approach I, the total element stiffness matrix can be written as

$$[K] = \int_{\Omega_0} \left[\left([B]^T + [B^*]^T \right) [D] \left([B] + [B^*] \right) + [B^{**}] \right] d\Omega_0 \tag{15}$$

where $[B]$ is the conventional geometric matrix for small deformations; $[D]$ is the conventional linear elastic modulus matrix; $[B^*]$ is a nonlinear term due to large deformation and is dependent

on the current deformation; and $[B^{**}]$ is an additional large-deformation contribution to $[K]$ and is dependent on the current stress state. The element force vectors are given by

$$\{f^1\} = \int_{\Gamma_0} [N]^T \{t\} d\Gamma - \int_{\Omega_0} \left([B]^T + [B^*]^T\right)\{S\} d\Omega_0$$
$$\{f^2\} = \int_{\Omega_0} \left([B]^T + [B^*]^T\right)[D]\{\Delta E^P\} d\Omega_0 \qquad (16)$$

where $[N]$ is the conventional shape function matrix; $\{t\}$ is the conventional element boundary surrace traction vector; and $\{S\}$ and $\{\Delta E^P\}$ are the corresponding vector forms of the stress tensor, S, and the plastic strain tensor, ΔE^P, respectively. It is noted that the surface traction term in Eqs. (16) is integrated over the element boundary Γ_0 of the reference configuration Ω_0. This is correct for all small-deformation problems. It is correct for large deformation problems only if both the magnitude and direction of the loads are independent of the deformation of the element, otherwise Γ_0 must be replaced by Γ of the current configuration Ω. Currently Eqs. (16) is used in ZIP2DL, which will allow the application of traction loads in terms of concentrated forces. However, pressure loading is not currently permitted under large-deformation conditions.

In Approach II, the total element stiffness matrix is obtained from Eq. (15) by simply replacing $[D]$ with its counterpart, $[D^{el\text{-}pl}]$, for elastic-plastic problems. The element force vector $\{f^1\}$ is the same as that in Eqs. (16), while the force vector $\{f^2\}$ disappears in Approach II.

Standard Gauss quadrature is used to integrate all stiffness matrices and force vectors. A reduced integration option is provided for plane strain problems where mesh locking may occur when Poisson's ration is close or equal to 0.5 or when incompressible plastic strains dominate elastic strains. This option is recommended when eight-noded quadrilateral elements are used in plane strain, in which case the reduced integration requires the use of 2 by 2 Gauss points.

2.3 FINITE ELEMENT SOLUTION METHODS

The resulting nonlinear finite element equations, Eq. (12) or (14), will be solved iteratively using the standard or modified Newton-Raphson method [4]. When Eq. (12) is used, it

is called ***Approach I*** in ZIP2DL, in which case the effect of plasticity is taken into account by treating it as an additional body force, leading to a modified force vector. On the other hand, when Eq. (14) is used, it is called ***Approach II***, in which case plasticity modifies the stiffness matrix instead of the force vector. When the standard Newton-Raphson method is used, the total stiffness matrix $[K]$ in either of the two approaches is updated in every iteration, while when a modified Newton-Raphson method is used the stiffness matrix is updated less frequently. It is noted that when Approach I is used for small-strain, small-rotation problems, $[K^2]$ will disappear from Eq. (12), and $[K]$ will be a constant, hence [K] does not need to be updated at all.

Equation (12) or (14) is solved one load step (defined in ZIP2DL as a time step) at a time. This solution procedure is started by setting the initial stress state to zero at the the beginning of the first load step and by setting the plastic strain increment to zero at the beginning of all load steps. A converged solution for each load step is obtained iteratively as follows: (a) At the start of every iteration, evaluate $[K]$ (if it needs to be updated) and $\{\Delta f\}$ based on the current loading, stress state and plastic strain increment (see Eqs. (15) and (16)); (b) Solve Eq. (12) or (14) for the displacement increment $\{\Delta U\}$ and update the displacement; (c) Calculate strain increment and update the strain; (d) Integrate the incremental stress-strain relations to obtain increments for stress and effective plastic strain; (e) Update stress, effective plastic strain and flow stress; (f) Check convergence; and (g) If convergence is reached, go to the next load step (or stop if this is the last load step), otherwise go to the next iteration by repeating the above procedure.

The plastic strain increment is used in step (a) only if Approach I is used, that is, only if the effect of plasticity is treated as an equivalent body force in the finite element equations in the form of a contribution to the force vector $\{\Delta f\}$. Because the plastic strain increment is unknown at the beginning of an iteration, it must be approximated. For example, the difference between the plastic strain at the beginning of the current iteration and the plastic strain at the beginning of the preceding iteration can be used for this purpose. However, a faster convergence is often achieved if the plastic strain increment is approximated by the plastic strain difference multiplied with a positive coefficient. This coefficient is called the ***acceleration multiplier***, which is usually

smaller than unity. At the moment, there is no formula that will automatically give the optimal value for this multiplier. Hence, this option is recommended for advanced users only.

Two different stress integration algorithms are made available. In the ***"path-dependent" algorithm***, the strain increment in step (c) is the increment from the preceding iteration to the current iteration. Hence stress is integrated from the stress level of the preceding iteration to the current iteration. In the ***"path-independent" algorithm***, the strain increment in step (c) is the total accumulated strain increment for the current load step. Hence stress is integrated from the stress level at the end of the last load step to the current iteration of the current load step. In general, solutions obtained with the path-independent algorithm is much more accurate than those with the path-dependent algorithm, hence larger load steps can be used with the former. The drawback of the path-independent algorithm is that it may take much more iterations to reach convergence unless a proper acceleration multiplier is used for the plastic strain increment.

Convergence of the iterative solution procedure for each load step is monitored and controlled by two error indicators. The first indicator is the ratio between the norm of $\{\Delta f\}$ (the out-of-balance force) at the current iteration to the value of the norm at the first iteration of the load step. The second indicator is the ratio between the inner product (the energy increment) of $\{\Delta f\}$ and $\{\Delta U\}$ at the current iteration and to the value of the inner product at the first iteration of the load step. Convergence is reached when the values of both error indicators are smaller than a user defined error tolerance denoted by TOL. It is worth noting that the force vectors in Eqs. (16) will each approach zero as convergence is reached.

2.4 ELASTIC-PLASTIC COMPUTATION ALGORITHMS

It is assumed in ZIP2DL that elastic-plastic materials obey the J_2 flow theory of plasticity, i.e., the von Mises yield criterion and the associated flow rule. Strain hardening is assumed to be isotropic. For brevity, discussions below use tensor notation and are for three-dimensional problems. Let $\mathbf{S}$ be the stress tensor, then the von Mises yield criterion (the yield condition) can be written in terms of its deviatoric portion, $\mathbf{S}^*$, as

$$F(\sigma,\varepsilon_p) \equiv \frac{3}{2} S^* : S^* - \sigma^2(\varepsilon_p) = 0 \tag{17}$$

where F is usually called the yield function and F=0 represents the current yield surface, and σ is the current flow (yield) stress which depends on the effective plastic strain ε_p. The associated flow rule is given by

$$d\varepsilon^p = d\lambda S^* \tag{18}$$

where $d\varepsilon^p$ is the plastic strain increment tensor. The plastic multiplier, $d\lambda$, must be nonnegative. Computationally, however, $d\lambda$ may become negative if flawed algorithms are used. This numerical phenomenon is called "***negative plastic flow***" and may cause severe convergence problems, especially for crack growth simulations that employ refined near-crack-tip meshes. As discussed in detail later, this problem is now clearly understood and a simple algorithm has been developed by Deng and Rosakis [7] that will prevent negative plastic flow from occurring.

The flow stress in Eq. (17) equals the initial yield stress (a constant) for elastic-perfectly plastic solids, and it is a function of the current effective plastic strain, ε_p, for strain hardening materials, where ε_p is defined by

$$\varepsilon_p = \int d\varepsilon_p, \text{ where } d\varepsilon_p = \sqrt{\frac{2}{3} d\varepsilon^p : d\varepsilon^p} \tag{19}$$

Equations (17)-(19) can be used to derive a relation between the effective plastic strain increment $d\varepsilon_p$ and $d\lambda$, which is given by

$$d\varepsilon^p = \frac{2}{3}\sigma d\lambda \tag{20}$$

which implies that once $d\lambda$ is caculated, $d\varepsilon_p$ can be evaluated and added to ε_p to update ε_p, which then can be used to update σ through its dependence on ε_p. Equation (20) also tells that when $d\lambda$ is negative, so is $d\varepsilon_p$ (hence the name "negative plastic flow"), which makes ε_p decreases and is physically wrong, because plastic deformation is irriversible and ε_p can not be decreased.

The dependence of the flow stress σ on the effective plastic strain ε_p is usually determined from uniaxial tension tests, in which case $\varepsilon_p=\varepsilon-\sigma/E$, where ε is the total normal strain

and E is the Young's modulus. Five types of material stress-strain models are available in ZIP2DL to approximate test data. They are:

(a) ***linearly elastic model***, in which no yielding will occur.

(b) ***elastic-perfectly plastic model***, in which $\sigma=\sigma_0$, the initial yield stress (also used in models below), and is a constant.

(c) ***power-law hardening model***, in which

$$\varepsilon_p = \frac{\sigma_0}{E}\left(\frac{\sigma}{\sigma_0}\right)^n - \frac{\sigma}{E} \tag{21}$$

where n (n>1) is the power-law hardening exponent.

(d) ***linear hardening (bi-linear) model***, in which the stress-strain curve is composed of an elastic line segment and a plastic line segment and the flow stress in the plastic line segment is

$$\sigma = \sigma_0 + \frac{\alpha E \varepsilon_p}{1-\alpha} \tag{22}$$

where α ($0\le\alpha<1$) is the linear hardening parameter, and is defined as the ratio between the slope (tangent modulus, E_t) of the plastic line and that (Young's modulus, E) of the elastic line. When $\alpha=0$ it is equivalent to the elastic-perfectly plastic model.

(e) ***multi-linear hardening model***, in which the stress-strain curve is approximated by a piece-wise linear curve, consisting of an elastic line segment and a number of plastic line segments. For plastic flow in the J-th plastic line segment, suppose the stress and strain at the lower end (the first end) of the segment is (σ_{J-1}, ε_{J-1}) and those at the higher end (the second end) of the segment is (σ_J, ε_J), then the flow stress is given by

$$\sigma = \frac{\sigma_{J-1}}{1-\alpha_J} + \frac{\alpha_J E}{1-\alpha_J}(\varepsilon_p - \varepsilon_{J-1}) \tag{23}$$

where $\alpha_J=E_{t\,J}/E$, with $E_{t\,J}$ being the tangent modulus of the J-th plastic line segment, given by

$$E_{tJ} = \frac{\sigma_J - \sigma_{J-1}}{\varepsilon_J - \varepsilon_{J-1}} \tag{24}$$

In general, $0\le\alpha_J<1$, where the zero value is used only for the ***last plastic line segment*** and only when it has a slope of zero.

For continued plastic deformation, Eq. (17) must be satisifed at all times, which requires that its differential must also be true at all times, leading to the following ***consistency condition***

$$3S^*:dS^* = 2\sigma\frac{d\sigma}{d\varepsilon_p}d\varepsilon_p \tag{25}$$

The consistency condition assures the continued satisfaction of the yield condition during an continuous plastic deformation if the yield condition is satisfied initially. It is noted that the consistency condition based on Eq. (25) is exact if the increments are infinitesimal and is called here the ***first-order interpretation*** of the consistency condition. In finite element computations, however, the stress and strain increments are not infinitesimal. Consequently, the yield condition at the end of a finite load step will not, in general, be satisfied. Conventionally, a ***radial return*** algorithm [11, 12] is employed to pull the stress state back to the yield surface. Alternatively, a ***higher-order interpretation*** of the consistency condition [13] can be used, in which Eq. (25) is improved by considering the effect of finite stress and strain increments. For elastic-perfectly plastic and linear hardening materials, the yield condition will be exactly satisfied at the end of a finite load step if the improved consistency condition is used. For other hardening materials, the yield condition can also be better satisfied.

For the purpose of discussion, the tensorial form of the ***tangent-stiffness based incremental stress-strain relations*** used in Eq. (10) are listed below

$$dS = C^{el-pl}:d\varepsilon, \quad \text{where } C^{el-pl} = \frac{(C:S^*)\otimes(C:S^*)}{\frac{2}{3}\sigma^2\left(\frac{2}{3}\frac{d\sigma}{d\varepsilon_p}+2\mu\right)} \tag{26}$$

where $\otimes$ is the tensor product symbol and μ, the shear modulus, and $d\lambda$ is given by

$$d\lambda = \frac{2\mu S^*:d\varepsilon}{\frac{2}{3}\sigma^2\left(\frac{2}{3}\frac{d\sigma}{d\varepsilon_p}+2\mu\right)} \tag{27}$$

Therefore, once a strain increment $d\varepsilon$ is calculated from a load step, the stress increment tensor dS and the plastic multiplier $d\lambda$ (hence the plastic strain increment tensor, from Eq. (18), and the effective plastic strain increment, from Eq. (20)) can be readily obtained.

As discussed earlier, $d\lambda$ must be nonnegative to be correct, but it can be computationally negative when evaluated from Eq. (27). The cause of this problem is that when a flawed algorithm is used to integrate stresses from strain increments (e.g. [14]), the inner product $S^*:d\varepsilon$ may become negative. As pointed out in [7], for methods not based on the tangent stiffness, negative plastic flow will not occur but an associated error will still exist without notice. The source of this and associated problems has been identified and a simple and effective remedy has been suggested [7] to improve existing algorithms.

A tangent-stiffness based stress integration algorithm, with the the remedy for preventing negative plastic flow, has been implemented in ZIP2DL. It combines tangent stiffness with radial return, and is called the ***tangent predictor-radial return method*** [15, 16]. This method can be outlined as follows.

Suppose $\mathbf{S}$ is the current stress state, ε_p is the current effective plastic strain and σ is the current flow stress, and suppose we wish to integrate a strain increment $\Delta\varepsilon$ to obtain a new stress state. First, assuming elastic behavior, a trial stress increment $\Delta\mathbf{S}^T=\boldsymbol{C}:\Delta\varepsilon$ and a trial stress state $\mathbf{S}^T=\mathbf{S}+\Delta\mathbf{S}^T$ are calculated. The trial stress state is then tested in the yield function $F(\mathbf{S}^T, \varepsilon_p)$. If $F<0$, the elastic assumption is valid and $\mathbf{S}^T$ is the new stress state. Otherwise the strain increment is partly in an elastic path and partly in a plastic path. The transition point, which represents a stress state in contact with the current yield surface, can be denoted by scalar $R\in[0, 1]$ and determined from $F(\mathbf{S}+R\Delta\mathbf{S}^T, \varepsilon_p)=0$. The contact stress state is then $\mathbf{S}^C=\mathbf{S}+R\Delta\mathbf{S}^T$ and lies exactly on the current yield surface. Compute the deviatoric stress tensor $\mathbf{S}^*$ from $\mathbf{S}^C$ and use $(1-R)\Delta\varepsilon$ as the strain increment. Plug in these quantities into Eq. (26) to obtain the stress increment $\Delta\mathbf{S}$ relative to $\mathbf{S}^C$, and into Eq. (27) to obtain $\Delta\lambda$. Calculate $\Delta\varepsilon_p$ from Eq. (20) and add $\Delta\varepsilon_p$ to ε_p to update ε_p, then update σ using its dependence on ε_p. Add $\Delta\mathbf{S}$ to $\mathbf{S}^C$ to form the new stress state $\mathbf{S}$ (***tangent prediction***). Test $\mathbf{S}$ in $F(\mathbf{S}, \varepsilon_p)$ with updated values for ε_p and σ. If $F\neq0$ then ***radial return*** is performed to scale $\mathbf{S}$ proportionally so that $F=0$.

To improve accuracy, ***subincrementation*** strategies are generally used to divide the portion of the strain increment that is in the plastic path into smaller increments (subincrements) and perform the above stress integration procedure for each of the subincrements in series. Two subincrementation options are available in ZIP2DL. One is based on a simple estimate using stresses, and the other based on a more elaborate scheme using strains. Usually the strain based estimate works better. In each case, a small positive number, TOLM (see Data Line 3 for the general input file zip2dl.in in the user manual), is used to make this estimate.

2.5 CRACK-GROWTH SIMULATION TECHNIQUES

ZIP2DL can be used for mixed-mode crack growth simulations with arbitrary crack growth paths. However, it cannot be used to predict the direction of crack growth; rather it asks the user to provide the crack growth path as input. As in ZIP2D, cracks are identified by finite element node pairs along crack faces. A node pair consists of two finite element nodes which reside on the opposite sides of the crack faces. These two nodes will occupy the same spatial location if the crack is sharp (e.g. fatigue cracks) or separated by the gap between the crack faces if the crack has a finite thickness (e.g. notch cracks). To facilitate crack growth simulation, "***crack paths***" are introduced in ZIP2DL to represent the original cracks plus their possible paths of extension. Similarly, these crack paths are identified by node pairs along these paths. If a node pair is on the unbroken portion of a crack path, then the two nodes of the pair will have the same coordinates, and they will be tied (bonded) together by two mutually perpendicular ***rigid*** springs to prevent them from separation before the crack cuts through them and breaks the bond.

Crack growth simulation is achieved through a crack-tip ***nodal release procedure*** that "unzips" the crack-tip node pair when a specified fracture criterion is satisifed. In ZIP2DL, the fracture criterion is checked at the end of every loading step and at the end of every crack growth step. When the criterion is met, the crack will advance one step along the crack growth path, as defined by the crack path. This is achieved by reducing the stiffness of the two rigid springs (that tie the crack-tip node pair together) to zero in a number of iterations, at the end of which the crack will have grown along the growth path by an amount equal to the size of the element ahead

of the crack tip. An alternative way of breaking the crack-tip node pair is to replace the springs for the node pair with the forces in the springs and reduce the forces to zero in a number of iterations. This second approach is used in ZIP2D, but is not currently implemented in ZIP2DL.

2.6 REFERENCES

1. Newman, J.C., Jr.: Finite-Element Analysis of Fatigue Crack Propagation Including the Effects of Crack Closure. Ph.D. Thesis, Virginia Polytechnic Institute, Blacksburg, VA, 1974.
2. Newman, J.C., Jr.: User guide for ZIP2D. NASA Langley Research Center, Hampton, Virginia, modified April 2, 1996.
3. Malvern, L.E.: *Introduction to the Mechanics of a Continuous Medium.* Prentice-Hall, Englewood Cliffs, New Jersey, 1969.
4. Bathe, K.-J.: *Finite Element Procedures.* Prentice-Hall, Englewood Cliffs, New Jersey, 1996.
5. Zienkiewicz, O.C.; Valliappan, S.; and King, I.P.: Elasto-Plastic Solutions of Engineering Problems: Initial Stress Finite Element Approach. *International Journal for Numerical Methods in Engineering*, vol. 1, 1969, pp. 75-100.
6. Marcal, P.V.: A Stiffness Method for Elastic-Plastic Problems. *Int. J. Mech. Sci.*, vol. 7, 1965, pp. 220-238.
7. Deng, X.; and Rosakis, A.J.: Negative Plastic Flow and its Prevention in Elasto-Plastic Finite Element Computation. *Finite Element in Analysis and Design*, vol. 7, 1990, pp. 181-191.
8. Newman, J.C., Jr.; Booth, B.C.; and Shivakumar, K.N.: An Elastic-Plastic Einite-Element Analysis of the J-Resistance Curve Using a CTOD Criterion. *Fracture Mechanics: Eighteenth Symposium, ASTM STP* 945, D.T. Read and R.P. Reed, Eds., American Society for Testing and Materials, Philadephia, 1988.
9. Sutton, M.A.; Zhao, W.; Dawicke, D.S.; Deng, X.; and Newman, J.C., Jr.: Numerical Investigations into the Viability of CTOD as a Fracture Parameter for Mixed-Mode I/II Tearing of Thin Aluminum Sheets. FAA-NASA Symposium on Continued Airworthiness of Aircraft Structures, Atlanta, August, 1996.

10. Deng, X.; and Newman, J.C., Jr.: Implementation and Application of a Large-Rotation Finite Element Formulation in NASA Code ZIP2DL. FAA-NASA Symposium on Continued Airworthiness of Aircraft Structures, Atlanta, August, 1996.
11. Wilkins, M.L.: Calculation of Elastic-Plastic Flow. *Methods of Computational Physics* Vol. 3, eds. B. Alder, S. Fernback and M. Roterneg, Academic Press, New York, 1964.
12. Krieg, R.D.; and Key, S.W.: Implementation of a Time-Independent Plasticity Theory into the Structural Computer Programs. *ASME AMD 20: Constitutive Equations in Viscoelasticity*, eds. J.A. Stricklin and K.J. Saczalski, ASME, 1976, pp. 125-137
13. Deng, X.: Dynamic Crack Propagation in Elastic-Plastic Solids. Ph.D. Dissertation, California Institute of Technology, Pasadena, CA, 1990.
14. Owen, D.R.J.; and Hinton, E.: Finite Element in Plasticity: Theory and Practice. Pineridge Press, Swansea, England, 1980.
15. Krieg, R.D.; and Krieg, D.B.: Accuracies of Numerical Solution Methods for the Elastic-Perfectly Plastic Model. *J. Pressure Vessel Technol.*, Vol. 99, 1977, pp. 510-515.
16. Schreyer, H.L.; Kulak, R.F.; and Kramer, J.M.: Accurate Numerical Solutions for Elastic-Plastic Models. *J. Pressure Vessel Technol.*, Vol. 102, 1979, pp. 226-234.

3. USER MANUAL

3.1 INTRODUCTION

The finite element formulations discussed in the theoretical manual have been implemented in ZIP2DL under plane stress, plane strain and mixed state of stress conditions. Mixed state of stress analysis is often performed in crack growth simulations, in which plane-strain conditions are imposed around the crack tip while plane-stress conditions are imposed elsewhere. Standard FORTRAN 77 computer language is used.

Conventional four- and eight-noded isoparametric quadrilateral elements and three-noded constant-strain triangular elements are provided (currently only one type of elements can be used for a particular problem). At the element level, element nodes must be numbered sequentially and in a counterclockwise manner.

Loading can be applied through surface tractions on element sides, concentrated forces or displacements at nodes, and special body forces as defined in the input. Currently the surface tractions must be independent of deformation for large-deformation problems. A ***reference magnitude*** and a ***time dependence*** is required as input for each load, so that the load at each time (load) step is given by MAGNITUDE*TIME-DEPENDENCE-FUNCTION-VALUE. Linear dependence is the ***default*** time dependence. A user subroutine, SUBROUTINE FTIME, is available to define custom time dependence functions for all loads. It is currently assumed that all surface tractions follow one time dependence, and all body forces follow another time dependence. However, concentrated forces and nodal displacements can each have their own custom time dependence.

Five types of material models are available: (1) linearly elastic, (2) elastic-perfectly plastic, (3) power-law hardening, (4) linear hardening (bi-linear), and (5) mutli-linear hardening. Model (5) is the most flexiable one, in which the effective stress-strain curve is represented by line segments. The first line segment (***elastic line segment***) describes the elastic behavior and all

other line segments (***plastic line segments***) describe elastic-plastic behavior. Hence Model (5) is often used to approximate actual effective stress-strain curves obtained from uniaxial tension tests. It is noted that model (4) is a special case of model (5).

Multiple cracks can be modeled and their growth simulated, provided that the crack growth paths are specified as input. A crack path is defined as the original crack plus its possible extension. Each crack path is identified by finite element node pairs along the crack path, and the crack tip must be located at one of the node pairs. Currently there are no crack link-up algorithms in ZIP2DL. Therefore the results are unpredictable when two cracks grow near each other and link up.

As can be seen from the line-by-line input explanation in Section 3.6, there are many job control options available in ZIP2DL. Some of the options are intended for advanced users and many can be used by all users. To assist beginners, recommended input values are given whenever appropriate. It is a good idea to try out the example problems provided in Section 4 first.

3.2 PROGRAM SIZE CONTROL

Program size is controlled by parameter MTOT, which is the length of an unnamed common block, COMMON A(MTOT), that allocates storage for most of the arrays in ZIP2DL. When cracks are involved, parameter MTOTCK is used to control a named common block, COMMON /CRACK1/B(MTOTCK), that allocates storage for arrays associated with cracks.

ZIP2DL will automatically check to see if the available computer storage (memory) initially set in the above two common blocks in ZIP2DL is larger than the required storage for the problem to be solved. When the available storage is smaller than the required storage, an error message will be written to the output file zip2dl.out, along with the required storage. The user must then increase the values for MTOT and/or MTOTCK and recompile ZIP2DL.

Double precision is used in all subroutines through the IMPLICIT statement. When single precision is preferred (e.g. for execution on CRAY supercomputers), this statement must be modified and parameter IPREC must be set to 1, so that array storage can be properly allocated.

3.3 INPUT DATA FILES

There are four input data files: zip2dl.in, zip2dl.msh, zip2dl.rs1, zip2dl.rs2 (note that the input as well as output file names must be in lower case in a unix environment).

zip2dl.in is the general input data file. It contains everything needed to run ZIP2DL except finite element mesh information. zip2dl.msh is the mesh input file. It contains nodal coordinates and element connectivities. These two files are the only two necessary when solutions to be obtained are started from zero loads.

zip2dl.rs1 and zip2dl.rs2 are restart files that are needed only when solutions to be obtained are based on those of a previous time step. These files contain complete solution information for a previous time step. In particular, zip2dl.rs1 contains data that do not change from one time step to another (hence they are "time-independent"), while zip2dl.rs2 contains data that may change from one step to another (hence they are "time-dependent"). When these two files are used, the general input file, zip2dl.in, must still be present because job control parameters are modified in zip2dl.in.

3.4 OUTPUT DATA FILES

There are five output data files: zip2dl.out, zip2dl.dat, zip2dl.pls, zip2dl.msg, zip2dl.rs1, zip2dl.rs2. The last two are the restart data files discussed in the previous section. While zip2dl.rs1 is generated only once, zip2dl.rs2 is updated at the end of every converged time step and just before the crack-tip advances, or updated as instructed in the input file.

zip2dl.out is the general output data file. It contains echos to the input data, storage information, job-control warning messages, crack growth data, concise stress and strain solutions for user selected elements, and displacement solutions for user selected nodes. Echos to mesh input data can be suppressed by the user by setting the job control parameter NPRNT to 0 or -1.

zip2dl.dat supplements zip2dl.out by listing detailed stress and strain solutions for user selected elements. zip2dl.pls supplements zip2dl.out by collecting the coordinates of all yielded Guass points, outlining the plastic zone.

zip2dl.msg is the output file that contains the run-time messages not included in zip2dl.out. Among them are loading status messages, iteration convergence status at each time step, current and total number of iterations, and the number of times that [*K*] is updated. Most run-time messages can be suppressed by setting the job control parameter NPRNT to 0.

3.5 LINE-BY-LINE INPUT EXPLANATION

GENERAL INPUT FILE: zip2dl.in

Data Line 1: TITLE

Brief problem description.

TITLE = A brief description of the problem being solved. TITLE can be up to 80 characters long but the first character must be blank.

Data Line 2: NSTART, NSGEN, MODE, IRI, IUPD, IAPP, NPRNT, NGELE, NGNOD, MSD, NODTOT

General job control parameters.

NSTART	= 0	Used without restart files zip2dl.rs1 and zip2dl.rs2;
	= 1	Used with restart files to obtain solutions for later time steps; Parameters TB, TE, DT (see Data Line 3) must be set properly;
	= 2	Used with restart files to obtain solutions for later time steps; TE, DT must be set properly (TB will be set by the program); **Or** used with re-start files to improve a converged solution (e.g. with a smaller TOL value); TE must be set to the time when the converged solution was obtained (TB and DT will be set by the program).
NSGEN	= 0	Generate restart files only at the end of the job;
	> 0	Generate restart files at the end of the first load step, then every NSGEN steps, and the last load step. That is, generate retart files at the end of the 1st, 2nd, 3rd, ... and last load step if NSGEN= 1; at the end of the 1st, 3rd, 5th, ... and last load step if NSGEN= 2; at the end of the 1st, 4th, 7th, ... and last load step if NSGEN= 3; and so on;
MODE	= 0	Perform data checking only; no FEM solution;
	= 1	Perform small-deformation, linear, elastic analysis;
	= 2	Perform large-deformation, linear, elastic analysis;
	= 3	Perform small-deformation, elastic-plastic analysis;
	= 4	Perform small-strain, large-rotation, elast-plastic analysis.
IRI	= 0	No reduced integration; *Recommended in general*;
	= 1	Perform reduced integration; *Recommended for 8-noded elements in plane strain.*
IUPD	< 0	[K] is updated at the first iteration and all those iterations of a time step whose number minus 1 is a multiple of IUPD; For example, [K] is updated at the 1st, 2nd, 3rd, ..., iterations if IUPD= -1; at the 1st, 3rd, 5th, ..., iterations if IUPD= -2;

at the 1st, 4th, 7th, ..., iterations if IUPD= -3;

and so on;

IUPD= -1 is recommended when [K] must be updated.

= 0 [K] is not updated at all (alway use the initial [K]);

Recommended when MODE=1 or 3 with IAPP=1;

> 0 [K] is updated at the first IUPD iterations for each load step;

For example, [K] is updated

at the 1st iteration if IUPD= 1;

at the 1st and 2nd iterations if IUPD= 2;

at the 1st, 2nd, and 3rd iterations if IUPD= 3;

and so on.

IAPP = 1 Approach I, the effect of plastic strain increment is taken into account with an equivalent body force;

Recommended for small-deformation problems;

= 2 APPROACH II, the effect of plastic strain increment is taken into account with a contribution to the stiffness matrix [K];

Recommended for large-deformation problems.

NPRNT =-1 Write run-time messages and final solution to output;

NPRNT= -1 is generally recommended;

= 0 Write only final solution to output;

= 1 Write element and node information, run-time messages and final solution to output.

NGELE = Number of element groups (NGELE<=100) for which stress and strain output will be written;

= 0 Stress and strain output will be written for all elements;

NGNOD = Number of node groups (NGNOD<=100) for which displacement output will be written;

= 0 Displacement output will be written for all nodes.

MSD = Number of cracks (0, 1, 2, etc.)

NODTOT = Total number of node pairs of all the crack paths.

NOTES: (1) When NSTART=1 is used to obtain solutions for later time steps, the calculation will start at the time specified by TB (see Data Line 3); When NSTART=2 is used, the calculation will start at the time when the restart files were generated.

(2) Reduced integration is recommended for plane strain problems with Poisson's ratio = 0.5 or with dominant incompressible plasticity; Currently available only for 8-noded quadrilateral elements.

Data Line 3: TB, TE, DT, TOL, TOLM, TOLP, TOLY, NSUB

Time step and tolerance control parameters.

TB, TE	=	Beginning and ending time, respectively.
DT	=	Time increment (it must be nonzero).
TOL	=	Tolerance (say 1.0E-6) for testing iteration convergence; *Smaller TOL values require more iterations for convergence.*
TOLM	=	Small positive value (say 1.0E-4) for determining strain subincrementation number during an elastic-plastic loading; *Smaller TOLM values require more computer CPU time.*
TOLP	=	Tolerance (say 1.0E-8) for testing negative plastic flow.
TOLY	=	Tolerance (say 1.0E-8) for testing yielding condition; *Small TOLY values may lead to "divide by zero" errors.*
NSUB	=	Allowable number (say 100) of iterations for each time step; *Small NSUB values may lead to premature program termination.*

Data Line 4: NUMN, NUMEL, NUMAT, NUMPW, NUMBL, NUMML, NUMSG, MN, NRST, IOPT

Mesh and material/geometry set control parameters.

NUMN	=	Total number of nodes.
NUMEL	=	Total number of elements.
NUMAT	=	Total number of material/geometry sets.
NUMPW	=	Total number of power-law hardening material sets.
NUMBL	=	Total number of bi-linear (linear hardening) material sets.
NUMML	=	Total number of multi-linear hardening material sets.
NUMSG	=	Total number of plastic line segments of all multi-linear material sets. For example, if we have two materials, one with 3 plastic line segments and the other with 6 segments, then NUMSG = 3+6 = 9.
MN	= 3	Mesh is composed of all 3-noded triangular elements;
	= 4	Mesh is composed of all 4-noded quadrilateral elements;
	= 8	Mesh is composed of all 8-noded quadrilateral elements.
NRST	= 1	Use 1X1 Gauss quadrature (Mainly for 3-noded elements);
	= 2	Use 2X2 Gauss quadrature (Quadrilateral elements only);
	= 3	Use 3X3 Gauss quadrature (Quadrilateral elements only).
IOPT	= 0	Plane strain calculation;
	= 1	Plane stress calculation;
	=-1	Plane stress with a fixed plane strain core;
	=-2	Plane stress with a moving plane strain core (Currently not available).

Data Line 5: NUMLD, NUMSS, NBODY, NDISP

Boundary condition control parameters.

NUMLD	=	Total number of degrees of freedom (DOFs) with specified nodal loads.
NUMSS	=	Total number of element sides with specified tractions.
NBODY	= 0	Body forces are absent;

	= 1	Body forces are present.
NDISP	= 0	No nonzero displacement is specified at any node;
	= 1	Nonzero displacements are specified at some nodes.

Data Line 6: NNC, NMC, NPD, NRR, ACC

Control parameters for plastic-flow factor calculation.

NNC	= 0	Use a first-order interpretation of the consistency condition for conventional plastic-flow factor calculation.
	= 1	Use a higher-order interpretation of the consistency condition for enhanced plastic-flow factor calculation.
NMC	= 0	Use a stress-based, simple estimate for the subincrement number in plastic strain calculation;
	= 1	Use a strain-based, elaborate estimate for the subincrementation number in plastic strain calculation;
NPD	= 0	Use "path-dependent" stress integration algorithm;
	= 1	Use "path-independent" stress integration algorithm;
NRR	= 0	Do not use "radial-return" stress integration algorithm;
	= 1	Use "radial-return" stress integration algorithm.
ACC	=	Scalar multiplier for plastic strain increments, used to accelerate numerical convergence when IAPP=1. *It is generally recommended for beginners to use NNC=0, NMC=1, NPD=0, NRR=1, ACC=1.0.*

Data Line 7: ESPR, NGRW, NLMT, NCRI, LCRI, NRLX, CTOD

Crack growth control parameters;

Skip this line if MSD=0.

ESPR	=	Rigid spring stiffness, equals the maximum Young's modulus of all materials multiplied by, say 1.0E+8.
NGRW	= 0	Cracks are stationary (no crack growth);
	= 1	Cracks will grow until the maximum load is reached;
	= 2	Cracks will grow until the maximum crack growth step number, NLMT, is reached by any of the cracks.
NLMT	=	Allowable number of crack growth steps for any crack.
NCRI	= 1	Critical CTOD criterion (currently only criterion available).
LCRI	=	Integer used for selecting location of fracture criterion evaluation;
	< 0	Fracture criterion is checked at the LCRI-th node behind the crack-tip (e.g. for criteria based on critical crack-tip-opening displacement behind the crack tip).
	= 0	Fracture criterion is checked at the crack tip (e.g for SIF and energy release rate criteria, currently not available);
	> 0	Fracture criterion is checked at the LCRI-th node ahead of the crack-tip (e.g. for criteria based on stresses and/or strains in front of the crack tip, currently not available).
NRLX	=	Number of iterations (3 is recommended) used to relax (release) a crack-tip node (rigid spring) when criterion for crack growth is satisfied.
CTOD	=	Critical crack-tip-opening displacement value.

NOTES: If divergence occurs after a nodal release when NRLX=3 is used, do the following: (a) stop the program; (b) save (copy and rename) the output and re-start files; (c) re-start the program with NSTART=2 and a larger value for NRLX (say 5).

Data Line 8: NELE(1), NELE(2), NELE(3)

Element groups for which stress and strain output will be written;

Currently limited to 100 groups (see COMMON /OUTPUT/ in the source code);
Repeat sequentially (DO I=1,NGELE);
Skip this line if NGELE=0.

NELE(1)	=	First element number for this element group.
NELE(2)	=	Last element number for this element group.
NELE(3)	=	Element number increment for this element group.

Data Line 9: NNOD(1), NNOD(2), NNOD(3)

Node groups for which displacement output will be written;
Currently limited to 100 groups (see COMMON /OUTPUT/ in the source code);
Repeat sequentially (DO I=1,NGNOD);
Skip this line if NGNOD=0.

NNOD(1)	=	First node number for this node group.
NNOD(2)	=	Last node number for this node group.
NNOD(3)	=	Node number increment for this node group.

Data Line 10: POIS, WT, THK, YM, SYI, MPAR(1)

Material/Geometry set data;
Repeat this line sequentially (DO I=1,NUMAT).

POIS	=	Poisson's ratio for the Ith material/geometry set.
WT	=	Mass density for the Ith set, used only with body forces.
THK	=	Plate thickness for the Ith set.
YM	=	Young's modulus for the Ith set.
SYI	=	Initial yield stress for the Ith set.
MPAR(1)	=-2	Linearly elastic material model;

=-1		Elastic-perfectly plastic material model;
= 0		Power-law hardening material model;
= 1		Bi-linear material model;
≥ 2		Multi-linear material model, where MPAR(1) is the number of plastic line segments in this model.

Data Line 11: M, HPAR

Hardening parameters for power-law material sets;
Repeat this line (DO I=1,NUMPW) for every set in this group;
Skip this line if NUMPW=0.

M	=	Material set number (integer 1, 2 or etc.).
HPAR	=	Exponent (n) for power-law hardening material set.

Data Line 12: M, HPAR

Hardening parameters for bi-linear (linear hardening) material sets;
Repeat this line (DO I=1,NUMBL) for every set in this group;
Skip this line if NUMBL=0.

M	=	Material set number (integer 1, 2 or etc.).
HPAR	=	Ratio of the slope (tangent modulus) of the plastic line segment to that (Young's modulus) of the elastic line segment (0<=HPAR<1).

Data Line 13: M, (HPAR(2*J-1), HPAR(2*J), J=1, MPAR(1)-1), HPAR(2*MPAR(1)-1)

Hardening parameters for multi-linear hardening material sets;
Repeat this line (DO I=1,NUMML) for every set in this group;
Skip this line if NUMML=0.

M	=	Material set number (integer 1, 2 or etc.).
MPAR(1)	=	Number of plastic line segments; Input in Data Line 10.
HPAR(2*J-1)	=	Stress at the end of the Jth plastic line segment.
HPAR(2*J)	=	Strain at the end of the Jth plastic line segment.
HPAR(2*MPAR(1)-1)	=	Ratio of the slope (tangent modulus) of the "last" plastic line segment to that (Young's modulus) of the elastic line segment (0<=HPAR(2*MPAR(1)-1)<1).

Data Line 14: (GRAV(I), I=1, 3)

Body force specification;
Skip this line if NBODY=0.

GRAV(1)	=	Body force per unit mass in x or radial direction.
GRAV(2)	=	Body force per unit mass in y or circumferential direction.
GRAV(3)	= 0	GRAR(1) and GRAV(2) are specified in a rectangular Cartesian system in x and y directions, respectively;
	= 1	GRAR(1) and GRAV(2) are specified in a polar system in radial and circumferential directions, respectively;
	= 2	Body forces are proportional to the radial distance, with GRAR(1) and GRAV(2) as the multiplier in the radial and circumferential directions, respectively.

Data Line 15: N, NE, NG, (ID1(I), I=1, 2)

Type of nodal degrees of freedom (DOF);
Input this line only for nodes with a specified (zero or nonzero) displacement in one or both directions;
Repeat this data line as often as needed;

End this data line with a new line of 5 zeros.

N, NE	=	Beginning and ending node numbers, respectively (NE>=N).
NG	=	Constant increment for node numbers from node N to node NE; (NG=0 when NE=N).
ID1(I)	= 0	Displacement is specified for the Ith DOF of all the nodes in this series.
	= 1	No displacement is specified for this DOF; This is the default value (set initially by the FEM program).

NOTES: The magnitudes of nonzero displacements are input in Data Line 17.

Data Line 16: FLD(I), NLTYPE(3*I-1), NLTYPE(3*I-2), NLTYPE(3*I)

Specified nodal loading (concentrated loads at nodal points).
Repeat this line (DO I=1,NUMLD) for all cases.
Skip this line if NUMLD=0.

FLD(I)	=	"Reference magnitude" for specified nodal load at the Ith DOF;
NLTYPE(3*I-1)=		Node number for the specified nodal load.
NLTYPE(3*I-2)=		Direction (1 for x and 2 for y) of the specified load.
NLTYPE(3*I) =		"Type of time dependence" of the specified load.

NOTES: See Section 3.1 (Introduction) and Section 3.6 (User Loading Subroutine) for definitions of "reference magnitude" and "type of time dependence."

Data Line 17: N, NE, NG, (R1(I), I=1, 2), (NR1(I), I=1, 2)

Specified nonzero nodal displacement loading;
Repeat this line as often as needed;

End this data line with 7 zeros;

Skip this line if NDISP=0.

N,NE	=	Beginning and ending node numbers, respectively (NE>=N).
NG	=	Constant increment for node numbers from node N to node NE; (NG=0 when NE=N).
R1(I)	=	"Reference magnitude" of specified nodal displacement for the Ith DOF; Use 0.0 if displacement is not specified at this DOF ;
NR1(I)	=	"Type of time dependence" of the specified displacement for the Ith DOF; Use 0 if displacement is not specified at this DOF.

NOTES: See Section 3.1 (Introduction) and Section 3.6 (User Loading Subroutine) for definitions of "reference magnitude" and "type of time dependence."

Data Line 18: IELNO(I), ISIDE(I), TX(I), TY(I), IT(I)

Specified boundary traction loading;

Repeat this line (DO I=1,NUMSS) for all cases.

Skip this line if NUMSS=0.

IELNO(I)	=	Element number with specified boundary traction.
ISIDE(I)	=	Element side number with specified boundary traction;
		FOR 3-NODED TRIANGULAR ELEMENTS:
	= 1	if traction is on the side between local nodes 1 and 2;
	= 2	if traction is on the side between local nodes 2 and 3
	= 3	if traction is on the side between local nodes 3 and 1.
		FOR 4- AND 8-NODED QUAdrilateral ELEMENTS:
	= 1	if traction is on the side between local nodes 1 and 2;
	= 2	if traction is on the side between local nodes 2 and 3
	= 3	if traction is on the side between local nodes 3 and 4.

= 4 if traction is on the side between local nodes 4 and 1.

TX(I) = "Reference magnitude" of specified boundary traction in the x direction if IT(I)=0, or in the normal direction if IT(I)=1;

TY(I) = "Reference magnitude" of specified boundary traction in th y direction if IT(I)=0, or in the tangential direction if IT(I)=1;

IT(I) = 0 or 1 as specified above.

NOTES: (1) The normal and tangential directions are defined for an observer who is walking along the sides of the element in the counterclockwise direction.
(2) See Section 3.1 (Introduction) and Section 3.6 (User Loading Subroutine) for definitions of "reference magnitude" and "type of time dependence."

****** The following lines are for crack growth problems.

****** Skip these lines if MSD=0.

Data Line 19: (LEN(I), I=1, MSD)

Number of node pairs for each crack path;
Skip this line if MSD=0.

LEN(I) = Total number of node pairs on the Ith crack path.

Data Line 20: (LTIP(1, I), LTIP(2, I), I=1, MSD)

Location of crack tips in the array of node pairs for each crack path;
Identified by the pair # of the crack-tip node pair in the array of crack-path node pairs;
Skip this line if MSD=0.

LTIP(1,I) = Location of the first crack tip of the Ith crack path;

=0, If this crack tip does not exist (for an edge crack).

LTIP(2,I) = Location of the second crack tip of the Ith crack path;

=0, If this crack tip does not exist (for an edge crack).

NOTES: Node pairs for each crack path are input sequentially in the array of node pairs from one end of the crack path to the other. Since cracks can extend from either crack tips, an end of a crack path is not necessarily a crack tip.

Data Line 21: NODP(1),NODP(2)

Node numbers of the crack-path node pairs;
Input data sequentially (First DO I=1,MSD; Then DO J=1,LEN(I));
Skip this line if MSD=0.

NODP(1) = Node number for the node on side #1 of the crack path.

NODP(2) = Node number for the node on side #2 of the crack path.

NOTES: (1) Since there are two nodes at every node on a crack path, one side of the crack path can be designated as side #1 while the other, side #2;
(2) If a crack has a nonzero thickness (e.g. a notch) then the two nodes of a node pair on the crack surface will have different coordinates;
(3) If a crack has a zero thickness (e.g. a fatigue crack) then the two nodes of a node pair on the crack surface will have the same coordinates;
(4) If a node pair is on the unbroken portion of a crack path then its two nodes will have the same coordinates;
(5) If a crack path is on a symmetry (or anti-symmetry) path and one half of the problem domain is removed, then use -1 (or -2) for NODP for the nodes on the side of the crack path that belongs to the removed portion of the problem domain;
(6) If a crack path is along the interface with a rigid material,

then use -3 for NODP for the nodes on the side of the rigid material.

To summarize:

NODP =-1 if the node is on a symmetric crack path;

=-2 if the node is on an anti-symmetric crack path;

=-3 if the node is on an interface with a rigid material;

where NODP is either NODP (1) or NODP(2)

depending on the side designation.

(7) Currently the orientation of symmetric or anti-symmetric crack paths can only be in the x direction.

Data Line 22: N, NE, NG

Plane strain core element numbers;

Repeat this data line as often as needed;

End this data line with a new line of 3 zeros;

Skip this data line if IOPT≠ -1.

N, NE	=	Beginning and ending element numbers (NE>=N), respectively.
NG	=	Constant increment for element numbers from element N to element NE (NG=0 when N=NE).

****** This is the end of the general input file.

***MESH INPUT FILE*:** zip2dl.msh

Data Line 1: N, NE, NG, MAT1, (LM1(I), I=1, NEN), LMG, NSP

Element numbering and element connectivity matrix;

Repeat this data line as often as needed;

NEN is the number of nodes for an element;
End this line with a new line of 9 zeros if NEN=3,
or 10 zeros if NEN=4, or 14 zeros if NEN=8.

N,NE	=	Beginning and ending element numbers (NE>=N), respectively.
NG	=	Constant increment for element numbers from element N to element NE (NG=0 when N=NE).
MAT1	=	Material/Geometry set number for the elements in this series.
LM1(I)	=	The global node # for the Ith local node of element N. NEN equals 3, 4 or 8. The local node numbers (1 to 3 for a 3-noded triangular element, or 1 to 4 for a 4-noded quadrilateral element, or 1 to 8 for an 8-noded quadrilateral element) are numbered sequentially in a counterclockwise fashion. For an 8-noded element, nodes 1 to 4 are the corner nodes and nodes 5 to 8 are the mid-point nodes on each of the four sides of the element, with node 5 between nodes 1 and 2, node 6 between nodes 2 and 3, and so on.
LMG	=	Constant increment for the connectivity matrix LM1(I) from element N to N+NG, ..., to NE (LMG=0 when N=NE).
NSP	= 0	if NEN=3, 4 (always) or if NEN=8 (in general);
	= 1	if NEN=8 and a special input format is used (for advanced users).

Data Line 2: N, NG, NBIAS, (COOR1(I), I=1, 2)

Node numbering and nodal coordinates;
Repeat this line as often as needed;
No need to input mid-node coordinates for 8-noded elements
(They will be generated automatically by the program);
End this data line with a new line of 5 zeros.

N	=	Beginning node number.

NG	=	Constant increment for node numbers from node N to node N', where N' is the node number of the next input line; NG=0 if coordinates are read one node at a time. All nodes from N to N' must be on a straight line.
NBIAS	=-1	Nodal spacing from N to N' decreases arithmatically;
	= 0	Nodal spacing from N to N' are uniform;
	= 1	Nodal spacing from N to N' increases arithmatically.
COOR1(I)	=	x coordinate of node N if I=1;
	=	y coordinate of node N if I=2.

****** This is the end of the mesh input file.

3.6 USER LOADING SUBROUTINE

A user subroutine, SUBROUTINE FTIME, is provided in ZIP2DL for defining custom loading histories (time dependence), and is located at the very end of the source code. In general, a load is defined as the product of a ***reference magnitude*** and a ***time-dependend*** function, whose value is given by the parameter FVAL in the arguments of the subroutine. For example, if a load has a linear time dependence and increases monotonically from zero to a maximum of 2.0, then one can chose the reference magnitude to be 2.0 and the time-dependent function to be the "time" T, and let T increases from 0.0 to 1.0, so that the load is given by 2.0*T. Alternatively, one can chose the reference magnitude to be 1.0 and the time-dependent function still to be T, but let T increases from 0.0. to 2.0, so that the load is simply given by T. The user should choose the reference magnitude and the time-dependent function that best fit his or her needs.

Because different types of loads may be present in a problem, it is desirable to allow these loads to have different loading histories. As such, a ***type*** is designated for the ***time dependence*** for each load. Currently, all body forces have one type of time dependence and is designated as NTYPE= -2, and all boundary tractions have one type and is defined by NTYPE= -1. However,

all concentrated nodal forces and nonzero nodal displacements will each have their own type, as defined by the user in the subroutine. A ***default*** of FVAL=T for the ***type of time dependence*** is given in the subroutine, where T is the time parameter in ZIP2DL.

The source code for the user loading subroutine is listed below.

```
      SUBROUTINE FTIME(NTYPE,T,FVAL)

C Defines the time dependence of various loadings.
C Load=Magnitude*FVAL, where Magnitude is defined in the
C input file and FVAL=F(T) is defined in this subroutine.

      IMPLICIT REAL*8(A-H,O-Z)
      COMMON /INFO1/NUMN,NUMEQ,NTOT,NPINFO,NUMTY,NUMLD,NBODY,NDISP
      COMMON /ELPAR/NUMEL,NUMAT,NUMSS,IOPT,NRST,MN

C Convention for loading types:
C NTYPE= -2...All body forces (Default for F(T): F(T)=T);
C      = -1...All boundary tractions (Default for F(T): F(T)=T);
C      => 1...As defined by the user for specified nodal loads
C             (concentrated forces) and nonzero displacements
C             (Default for F(T): F(T)=T).

C Set default time dependence.
      FVAL=T

C Set time dependence for all body forces.
      IF (NTYPE.EQ.-2)THEN
        FVAL=T     ! Default

C Set time dependence for all boundary tractions.
      ELSE IF(NTYPE.EQ.-1)THEN
        FVAL=T     ! Default

C Set time dependence for concentrated nodal forces and
C nonzero displacements, as defined below by the user.
      ELSE IF(NTYPE.GE.1)THEN
        FVAL=T     ! Default
      ENDIF
      RETURN
      END
```

4. EXAMPLE MANUAL

ZIP2DL has been calibrated with a variety of test problems involving all available combinations of material models, stress states (plane stress, plane strain, and mixed-state of stress), element types, kinematics relations, loading conditions, and solution approaches. Calibration solutions are obtained either analytically or numerically with established finite element codes. In the following, some of the test problems are provided as examples to illustrate the usage of ZIP2DL.

4.1 ELASTIC-PLASTIC TEST PROBLEMS

Elastic-plastic test problems include uniaxial tension-compression, biaxial tension-compression, and pure shear under plane stress and plane strain conditions, and for linear hardening, power-law hardening, multilinear hardening, and/or elastic-perfectly plastic materials.

The problem illustrated here is for the case of uniaxial tension in plane stress for a linear hardening material. Stresses are normalized by the initial yield stress and strains by the initial yield strain, so that the Young's modulus and the initial yield stress and strain can all be considered to be unity. The Poission's ratio is set to 0.3. The ratio between the slope (tangent modulus) of the plastic line segment to that (the Young's modulus) of the elastic line segment is 0.5. One four-noded, unit-square element is used. Node 1 is located at (1, 1), node 2 at (0, 1), node 3 at (0, 0), and node 4 at (1, 0). Traction loading is is applied in the normal (y) direction on side #1 (between nodes 1 and 2) of the element. The applied traction is started at zero and is then monotonically increased to 2.0 in ten steps using the default time dependence in the user subroutine. To remove rigid-body motions, symmetry conditions are enforced at nodes 3 and 4, so that node 3 is fixed while node 4 can only move horizontally. The element thickness does not play a role here but is set to 1.0.

For comparison, the analytical solution is given here. Suppose the applied normal traction is denoted by σ, then the effective stress σ_{eff} and the vertical normal stress σ_y are equal to σ, and

the horizontal normal stress σ_x is zero. For $\sigma_{eff} \leq \sigma_o$, the normal strains are given by $\varepsilon_y = \sigma/E$ and $\varepsilon_x = \varepsilon_z = -\nu\sigma/E$, where σ_o is the initial yield stress, ν is the Poisson's ratio and E is the Young's modulus. For $\sigma_{eff} \geq \sigma_o$, then $\varepsilon_y = \sigma/E + \varepsilon_p$, and $\varepsilon_x = \varepsilon_z = -\nu\sigma/E - \varepsilon_p/2$, where ε_p is the effective plastic strain and is given by $\varepsilon_p = (1-\alpha)(\sigma-\sigma_o)/(\alpha E)$, where α is the linear hardening parameter, which is the ratio of the tangent modulus, E_t, of the plastic line segment to the Young's modulus E. For the test problem illustrated here, E=1.0, σ_o=1.0, and α=0.5. Hence, when the loading reaches σ=2.0, the solutions for the strains are ε_p=1.0, ε_y=3.0, and $\varepsilon_x = \varepsilon_z$= -1.1.

The general input file zip2dl.in is listed below:

```
Uniaxial tension in plane stress with linear hardening
0,1,3,0,2,2,1,0,0,0,0
0.1,1.0,.1,1.e-6,1.e-4,1.e-8,1.e-8,100
4,1,1,0,1,0,0,4,2,1
0,1,0,0
0,1,0,1,1.0
.3,0.,1.,1.e+0,1.e+0,1
1,0.5
3,3,0,0,0
4,4,0,1,0
0,0,0,0,0
1,1,0.,2.,0
```

The mesh input file zip2dl.msh is listed below:

```
1,1,0,1,1,2,3,4,0,0
0,0,0,0,0,0,0,0,0,0
1,0,0,1.,1.
2,0,0,0.,1.
3,0,0,0.,0.
4,0,0,1.,0.
0,0,0,0.,0.
```

The general output file zip2dl.out is listed below:

```
Uniaxial tension in plane stress with linear hardening

NSTART=  0   NSGEN=  1   MODE =  3   IRI= 0   IUPD  =   2   IAPP = 2
NPRNT =  1   NGELE=  0   NGNOD=  0   MSD= 0   NODTOT=   0

TB  = 0.100E+00  TE  = 0.100E+01  DT  = 0.100E+00  TOL = 0.100E-05
TOLM= 0.100E-03  TOLP= 0.100E-07  TOLY= 0.100E-07  NSUB=  100
```

```
NUMN =     4  NUMEL=     1  NUMAT= 1
NUMPW=     0  NUMBL=     1  NUMML= 0  NUMSG= 0  MN= 4  NRST= 2  IOPT= 1

NUMLD=     0  NUMSS=     1  NBODY= 0  NDISP= 0

NNC  =     0  NMC  =     1  NPD  = 0  NRR  = 1  ACC = 1.00

MATERIAL/GEOMETRIC SETS
  SET     POIS         WT          THK         YM          SYI      MPAR(1)    MPAR(2)
   1  0.300E+00  0.000E+00  0.100E+01  0.100E+01  0.100E+01     1          0

Bi-linear material group:
GLOBAL SET #      LINE MODULUS RATIO
      1                 0.50000

MATERIAL SET NUMBERS AND CONNECTIVITY VECTORS
      ELE   SET       NODE NUMBERS
        1     1         1      2      3      4


NODAL COORDINATES
      NODE          X              Y
        1     0.10000E+01    0.10000E+01
        2     0.00000E+00    0.10000E+01
        3     0.00000E+00    0.00000E+00
        4     0.10000E+01    0.00000E+00


NODAL DISPL     NODE  DOF     TYPE
 0.00000E+00      3    1       0
 0.00000E+00      3    2       0
 0.00000E+00      4    2       0


EQUATION NUMBERS
      NODE    DOF 1    DOF 2
        1       1        2
        2       3        4
        3       0        0
        4       5        0

BOUNDARY STRESS SETS
     SET      ELE    SIDE            TX              TY
       1        1       1     0.00000E+00     0.20000E+01


       ***** ELEMENT COMPUTATIONS START HERE *****


REQUIRED STORAGE FOR THE STIFFNESS MATRIX =      15

TOTAL REQUIRED STORAGE =     352   AVAILABLE STORAGE =  200000

    FOR ELEMENT     1:      AREA =  0.10000E+01

____SOLUTION AT T =  0.10000E+01 IS CONVERGED____
    Re-start files now only contain data for this time step;

    Number of iteration for this time step is    2;
    Total number of iterations so far is     21;
```

```
     [K] is updated    2 times in this time step;
     [K] is updated a total of     18 times so far.

 2ND P-K STRESSES & G-L NORMAL STRAINS:
 ELE #    COORD X   COORD Y  STRESS X  STRESS Y STRESS XY  STRAIN X  STRAIN Y
     1  0.789E+00 0.789E+00 0.397E-08 0.200E+01 -.329E-17 -.110E+01 0.300E+01
     1  0.211E+00 0.789E+00 0.397E-08 0.200E+01 -.145E-16 -.110E+01 0.300E+01
     1  0.211E+00 0.211E+00 0.397E-08 0.200E+01 -.615E-17 -.110E+01 0.300E+01
     1  0.789E+00 0.211E+00 0.397E-08 0.200E+01 0.204E-17 -.110E+01 0.300E+01

 DISPLACEMENTS:
 NODE#      COORD X     COORD Y     DISPL X     DISPL Y     DISPL R     DISPL T
     1   0.1000E+01  0.1000E+01 -0.1099E+01  0.3000E+01  0.1344E+01  0.2899E+01
     2   0.0000E+00  0.1000E+01  0.2764E-16  0.3000E+01  0.3000E+01 -0.2764E-16
     3   0.0000E+00  0.0000E+00  0.0000E+00  0.0000E+00
     4   0.1000E+01  0.0000E+00 -0.1099E+01  0.0000E+00 -0.1099E+01  0.0000E+00
```

REMARKS:

(1) Under small-deformation conditions, the 2nd P-K (Piola-Kirschhoff) stresses are equal to the Cauchy (true) stresses, and the G-L (Green-Lagrange) strains are equal to the engineering strains. Refer to reference [3] in the theoretical manual manual for detailed discussions about various stress and strain measures.

(2) Stress and strain solutions are given at Gauss integration (quadrature) points, so that if four Gauss points are used for an element, then solutions will be listed for all four points.

(3) In the output for stress, strain and displacement variations, "R" is used to stand for radial variations while "T" for tangential or circumferential variations.

The detailed stress and strain output file zip2dl.dat is listed below:

```
****** ELEMENT NUMBER     1  ******

***Cauchy Stresses:
POINT         X COOR        Y COOR      STRESS X     STRESS Y     STRESS XY
  1      0.78868E+00  0.78868E+00  0.39707E-08  0.20000E+01 -0.32899E-17

Principal Cauchy Stresses:
           PSTRESS 1    PSTRESS 2        ANGLE       PSHEAR
         0.20000E+01  0.39707E-08 -0.90000E+02  0.10000E+01

Green-Lagrange Strains:
           STRAIN  X    STRAIN  Y    STRAIN XY  STRS/STRN Z
        -0.10993E+01  0.30000E+01 -0.69268E-17 -0.11007E+01

Cauchy Stresses and Green-Lagrange Strains in Polar Coord System:
             R COOR       T COOR     STRESS R     STRESS T     STRESS RT
         0.11154E+01  0.45000E+02  0.10000E+01  0.10000E+01  0.10000E+01

           STRAIN  R    STRAIN  T    STRAIN RT
         0.95034E+00  0.95034E+00  0.40993E+01
```

```
EFFECTIVE STRESS = 0.20000E+01  YIELD SURFACE = 0.20000E+01
EFFECTIVE PLASTIC STRAIN = 0.10000E+01


***Cauchy Stresses:
POINT          X COOR        Y COOR      STRESS X      STRESS Y     STRESS XY
  2       0.21132E+00   0.78868E+00   0.39707E-08   0.20000E+01 -0.14468E-16

Principal Cauchy Stresses:
           PSTRESS 1     PSTRESS 2         ANGLE        PSHEAR
         0.20000E+01   0.39707E-08 -0.90000E+02   0.10000E+01

Green-Lagrange Strains:
           STRAIN  X     STRAIN  Y     STRAIN XY  STRS/STRN Z
        -0.10993E+01   0.30000E+01 -0.49260E-16 -0.11007E+01

Cauchy Stresses and Green-Lagrange Strains in Polar Coord System:
              R COOR        T COOR      STRESS R      STRESS T     STRESS RT
          0.81650E+00   0.75000E+02   0.18660E+01   0.13397E+00   0.50000E+00

            STRAIN  R     STRAIN  T     STRAIN RT
          0.27254E+01 -0.82471E+00   0.20497E+01

EFFECTIVE STRESS = 0.20000E+01  YIELD SURFACE = 0.20000E+01
EFFECTIVE PLASTIC STRAIN = 0.10000E+01


***Cauchy Stresses:
POINT          X COOR        Y COOR      STRESS X      STRESS Y     STRESS XY
  3       0.21132E+00   0.21132E+00   0.39707E-08   0.20000E+01 -0.61528E-17

Principal Cauchy Stresses:
           PSTRESS 1     PSTRESS 2         ANGLE        PSHEAR
         0.20000E+01   0.39707E-08 -0.90000E+02   0.10000E+01

Green-Lagrange Strains:
           STRAIN  X     STRAIN  Y     STRAIN XY  STRS/STRN Z
         -0.10993E+01   0.30000E+01 -0.18224E-16 -0.11007E+01

Cauchy Stresses and Green-Lagrange Strains in Polar Coord System:
              R COOR        T COOR      STRESS R      STRESS T     STRESS RT
          0.29886E+00   0.45000E+02   0.10000E+01   0.10000E+01   0.10000E+01

            STRAIN  R     STRAIN  T     STRAIN RT
          0.95034E+00   0.95034E+00   0.40993E+01

EFFECTIVE STRESS = 0.20000E+01  YIELD SURFACE = 0.20000E+01
EFFECTIVE PLASTIC STRAIN = 0.10000E+01


***Cauchy Stresses:
POINT          X COOR        Y COOR      STRESS X      STRESS Y     STRESS XY
  4       0.78868E+00   0.21132E+00   0.39707E-08   0.20000E+01   0.20410E-17

Principal Cauchy Stresses:
           PSTRESS 1     PSTRESS 2         ANGLE        PSHEAR
         0.20000E+01   0.39707E-08   0.90000E+02   0.10000E+01

Green-Lagrange Strains:
           STRAIN  X     STRAIN  Y     STRAIN XY  STRS/STRN Z
```

```
        -0.10993E+01   0.30000E+01   0.67573E-17 -0.11007E+01

 Cauchy Stresses and Green-Lagrange Strains in Polar Coord System:
              R COOR        T COOR      STRESS R      STRESS T     STRESS RT
           0.81650E+00   0.15000E+02   0.13397E+00   0.18660E+01   0.50000E+00

             STRAIN  R     STRAIN  T     STRAIN RT
          -0.82471E+00   0.27254E+01   0.20497E+01

 EFFECTIVE STRESS = 0.20000E+01  YIELD SURFACE = 0.20000E+01
 EFFECTIVE PLASTIC STRAIN = 0.10000E+01
```

REMARKS:

(1) "ANGLE" in the principal stress output is the angle between x-axis and the first principal stress direction.

(2) "YIELD SURFACE" in the output refers the current flow stress (yield stress). For a Gauss point that is currently yielded, the effective stress must be on the current yield surface.

The plastic zone output file zip2dl.pls is listed below:

```
PLASTIC ZONE GIVEN BY YIELDED GAUSS INTEGRATION POINTS:
 (The effective stress is within 2% of the flow stress)

      X-COOR          Y-COOR          EFF-STS         FLOW-STS      PT #      ELE #
 0.78868E+00     0.78868E+00     0.20000E+01     0.20000E+01         1          1
 0.21132E+00     0.78868E+00     0.20000E+01     0.20000E+01         2          1
 0.21132E+00     0.21132E+00     0.20000E+01     0.20000E+01         3          1
 0.78868E+00     0.21132E+00     0.20000E+01     0.20000E+01         4          1
```

The run-time message output file zip2dl.msg is listed below:

```
 THE BEGINNING OF A NEW TIME STEP: T =  0.10000E+00.
     Relative errors for force and energy increments: 0.100E+01 & 0.100E+01.
     Relative errors for force and energy increments: 0.649E-33 & 0.652E-33.

 ____SOLUTION AT T =  0.10000E+00 IS CONVERGED____
     Re-start files now only contain data for this time step;

     Number of iteration for this time step is    2;
     Total number of iterations so far is      2;

     [K] is updated    2 times in this time step;
     [K] is updated a total of      2 times so far.

     Max. Effective Stress =  0.20000E+00 at Gauss Pt 1 of Element      1
     Max. Relative Eff Strs = 0.200000E+00 at Gauss Pt 1 of Element      1
     (Maximum when divided by initial yield stress)
     (Elastic elements excluded)

     Reactions at nodes with specified displacements:
     NODE#       X FORCE       Y FORCE
```

```
        3   0.27217E-17  -0.10000E+00
        4  -0.14707E-17  -0.10000E+00

THE BEGINNING OF A NEW TIME STEP: T =  0.20000E+00.
    Relative errors for force and energy increments: 0.100E+01 & 0.100E+01.
    Relative errors for force and energy increments: 0.649E-33 & 0.652E-33.

____SOLUTION AT T =  0.20000E+00 IS CONVERGED____
    Re-start files now only contain data for this time step;

    Number of iteration for this time step is     2;
    Total number of iterations so far is      4;

    [K] is updated    2 times in this time step;
    [K] is updated a total of      4 times so far.

    Max. Effective Stress =  0.40000E+00 at Gauss Pt 1 of Element      1
    Max. Relative Eff Strs = 0.400000E+00 at Gauss Pt 1 of Element      1
    (Maximum when divided by initial yield stress)
    (Elastic elements excluded)

    Reactions at nodes with specified displacements:
    NODE#       X FORCE       Y FORCE
        3   0.27217E-17  -0.20000E+00
        4  -0.14707E-17  -0.20000E+00

THE BEGINNING OF A NEW TIME STEP: T =  0.30000E+00.
    Relative errors for force and energy increments: 0.100E+01 & 0.100E+01.
    Relative errors for force and energy increments: 0.639E-33 & 0.577E-33.

____SOLUTION AT T =  0.30000E+00 IS CONVERGED____
    Re-start files now only contain data for this time step;

    Number of iteration for this time step is     2;
    Total number of iterations so far is      6;

    [K] is updated    2 times in this time step;
    [K] is updated a total of      6 times so far.

    Max. Effective Stress =  0.60000E+00 at Gauss Pt 1 of Element      1
    Max. Relative Eff Strs = 0.600000E+00 at Gauss Pt 1 of Element      1
    (Maximum when divided by initial yield stress)
    (Elastic elements excluded)

    Reactions at nodes with specified displacements:
    NODE#       X FORCE       Y FORCE
        3   0.26359E-17  -0.30000E+00
        4  -0.16768E-17  -0.30000E+00

THE BEGINNING OF A NEW TIME STEP: T =  0.40000E+00.
    Relative errors for force and energy increments: 0.100E+01 & 0.100E+01.
    Relative errors for force and energy increments: 0.380E-33 & 0.460E-33.

____SOLUTION AT T =  0.40000E+00 IS CONVERGED____
    Re-start files now only contain data for this time step;

    Number of iteration for this time step is     2;
    Total number of iterations so far is      8;

    [K] is updated    2 times in this time step;
    [K] is updated a total of      8 times so far.

    Max. Effective Stress =  0.80000E+00 at Gauss Pt 1 of Element      1
    Max. Relative Eff Strs = 0.800000E+00 at Gauss Pt 1 of Element      1
    (Maximum when divided by initial yield stress)
    (Elastic elements excluded)
```

```
    Reactions at nodes with specified displacements:
    NODE#       X FORCE       Y FORCE
        3   0.24148E-17  -0.40000E+00
        4  -0.12472E-17  -0.40000E+00

THE BEGINNING OF A NEW TIME STEP: T =  0.50000E+00.
    Relative errors for force and energy increments: 0.100E+01 & 0.100E+01.
    Relative errors for force and energy increments: 0.380E-33 & 0.212E-17.

____SOLUTION AT T =  0.50000E+00 IS CONVERGED____
    Re-start files now only contain data for this time step;

    Number of iteration for this time step is    2;
    Total number of iterations so far is     10;

    [K] is updated    2 times in this time step;
    [K] is updated a total of     10 times so far.

    Max. Effective Stress =  0.10000E+01 at Gauss Pt 1 of Element     1
    Max. Relative Eff Strs = 0.100000E+01 at Gauss Pt 1 of Element     1
    (Maximum when divided by initial yield stress)
    (Elastic elements excluded)

    Reactions at nodes with specified displacements:
    NODE#       X FORCE       Y FORCE
        3   0.24148E-17  -0.50000E+00
        4  -0.12472E-17  -0.50000E+00

THE BEGINNING OF A NEW TIME STEP: T =  0.60000E+00.
    Relative errors for force and energy increments: 0.100E+01 & 0.100E+01.
    Relative errors for force and energy increments: 0.685E-05 & 0.195E-05.
    Relative errors for force and energy increments: 0.263E-07 & 0.305E-07.

____SOLUTION AT T =  0.60000E+00 IS CONVERGED____
    Re-start files now only contain data for this time step;

    Number of iteration for this time step is    3;
    Total number of iterations so far is     13;

    [K] is updated    2 times in this time step;
    [K] is updated a total of     12 times so far.

    Max. Effective Stress =  0.12000E+01 at Gauss Pt 1 of Element     1
    Max. Relative Eff Strs = 0.119997E+01 at Gauss Pt 1 of Element     1
    (Maximum when divided by initial yield stress)
    (Elastic elements excluded)

    Reactions at nodes with specified displacements:
    NODE#       X FORCE       Y FORCE
        3  -0.32645E-05  -0.59999E+00
        4   0.32645E-05  -0.59999E+00

    Max. and min. subincrement numbers: MCMAX=  1751 & MCMIN=   250.

THE BEGINNING OF A NEW TIME STEP: T =  0.70000E+00.
    Relative errors for force and energy increments: 0.100E+01 & 0.100E+01.
    Relative errors for force and energy increments: 0.813E-11 & 0.228E-11.

____SOLUTION AT T =  0.70000E+00 IS CONVERGED____
    Re-start files now only contain data for this time step;

    Number of iteration for this time step is    2;
    Total number of iterations so far is     15;

    [K] is updated    2 times in this time step;
    [K] is updated a total of     14 times so far.
```

```
     Max. Effective Stress =  0.14000E+01 at Gauss Pt 1 of Element      1
     Max. Relative Eff Strs = 0.140000E+01 at Gauss Pt 1 of Element      1
     (Maximum when divided by initial yield stress)
     (Elastic elements excluded)

     Reactions at nodes with specified displacements:
     NODE#        X FORCE        Y FORCE
         3    0.22123E-06  -0.70000E+00
         4   -0.22123E-06  -0.70000E+00

     Max. and min. subincrement numbers: MCMAX=  2000 & MCMIN=      1.

THE BEGINNING OF A NEW TIME STEP: T =  0.80000E+00.
     Relative errors for force and energy increments: 0.100E+01 & 0.100E+01.
     Relative errors for force and energy increments: 0.227E+00 & 0.510E+00.
     Relative errors for force and energy increments: 0.138E-04 & 0.823E-05.
     Relative errors for force and energy increments: 0.271E-09 & 0.556E-09.

_____SOLUTION AT T =  0.80000E+00 IS CONVERGED_____
     Re-start files now only contain data for this time step;

     Number of iteration for this time step is     4;
     Total number of iterations so far is      19;

     [K] is updated     2 times in this time step;
     [K] is updated a total of      16 times so far.

     Max. Effective Stress =  0.16000E+01 at Gauss Pt 1 of Element      1
     Max. Relative Eff Strs = 0.160000E+01 at Gauss Pt 1 of Element      1
     (Maximum when divided by initial yield stress)
     (Elastic elements excluded)

     Reactions at nodes with specified displacements:
     NODE#        X FORCE        Y FORCE
         3   -0.95674E-06  -0.80000E+00
         4    0.95674E-06  -0.80000E+00

     Max. and min. subincrement numbers: MCMAX=  1020 & MCMIN=      2.

THE BEGINNING OF A NEW TIME STEP: T =  0.10000E+01.
     Relative errors for force and energy increments: 0.100E+01 & 0.100E+01.
     Relative errors for force and energy increments: 0.164E-15 & 0.458E-16.

_____SOLUTION AT T =  0.10000E+01 IS CONVERGED_____
     Re-start files now only contain data for this time step;

     Number of iteration for this time step is     2;
     Total number of iterations so far is      21;

     [K] is updated     2 times in this time step;
     [K] is updated a total of      18 times so far.

     Max. Effective Stress =  0.20000E+01 at Gauss Pt 4 of Element      1
     Max. Relative Eff Strs = 0.200000E+01 at Gauss Pt 4 of Element      1
     (Maximum when divided by initial yield stress)
     (Elastic elements excluded)

     Reactions at nodes with specified displacements:
     NODE#        X FORCE        Y FORCE
         3   -0.19854E-08  -0.10000E+01
         4    0.19854E-08  -0.10000E+01

     Max. and min. subincrement numbers: MCMAX=  4001 & MCMIN=      1.
```

4.2 LARGE-DEFORMATION TEST PROBLEMS

Large-deformation test problems are performed under elastic conditions only. They include large-strain uniaxial tension-compression, biaxial tension-compression, and pure shear, all under both plane stress and plane strain conditions. For the case of small-strain and large-rotation, a rigid-body rotation problem is tested with several meshes.

The problem illustrated below is the arbitrary rigid-body rotation of a unit square. This square is divided into four equal square quadrilateral elements, with the four corner nodes numbered 1, 2, 3, 4. The coordinates of the four corner nodes are the same as those used for the elastic-plastic test problem discussed in the preceding section. The coordinates of other nodes are as follows. Node 5 is at (0.5, 1.0), node 6 at (0.0, 0.5), node7 at (0.5, 0.0), node 8 at (1.0, 0.5), and node 9 at (0.5, 0.5). The sqare is made to rotate freely around node 3 by specifying the x and y displacements at node 2 according to rigid-body rotation displacement formulus for that node. For the case shown here, the angle of rigid-body rotation is increased gradually from 0^0 to 140^0 (2.4435 radians) with an increment of 0.1 radians. The numerical solutions are obtained using ZIP2DL with a large-deformation option (either MODE=2 or 4). The material properties are the same as those for the elastic-plastic test problem except that the linear hardening materila is now represented by a multilinear hardening model by dividing the plastic line segment into 3 pieces.

For comparison, the exact solutions for the displacement components are given here. They are: $u_x=\cos(\theta+\pi/4)-1$, $u_y=\sin(\theta+\pi/4)-1$ at node 1 and $u_x=\cos(\theta)-1$, $u_y=\sin(\theta)$ at node 4, where θ is the angle of rotation in radians.

The general input file zip2dl.in is listed below.

```
Large rigid-body rotation test
0,1,4,0,-1,2,1,0,0,0,0
0.1,2.443,.1,1.e-6,1.e-4,1.e-8,1.e-8,100
9,4,1,0,0,1,3,4,2,1
0,0,0,1
0,1,0,1,1.0
.3,0.,1.,1.e+0,1.e+0,3
1,1.2,1.4,1.3,1.6,0.5
```

```
3,3,0,0,0
2,2,0,0,0
0,0,0,0,0
2,2,0,1.,1.,2,3
0,0,0,0.,0.,0,0
```

The mesh input file zip2dl.msh is listed below.

```
1,1,0,1,1,5,9,8,0,0
2,2,0,1,5,2,6,9,0,0
3,3,0,1,9,6,3,7,0,0
4,4,0,1,8,9,7,4,0,0
0,0,0,0,0,0,0,0,0,0
1,1,0,1.,1.
2,1,0,0.,1.
3,1,0,0.,0.
4,0,0,1.,0.
5,1,0,.5,1.
6,1,0,0.,.5
7,1,0,.5,0.
8,1,0,1.,.5
9,1,0,.5,.5
0,0,0,0.,0.
```

The user loading subroutine is written as:

```
      SUBROUTINE FTIME(NTYPE,T,FVAL)

C Defines the time dependence of various loadings.
C Load=Magnitude*FVAL, where Magnitude is defined in the
C input file and FVAL=F(T) is defined in this subroutine.
      IMPLICIT REAL*8(A-H,O-Z)
      COMMON /INFO1/NUMN,NUMEQ,NTOT,NPINFO,NUMTY,NUMLD,NBODY,NDISP
      COMMON /ELPAR/NUMEL,NUMAT,NUMSS,IOPT,NRST,MN

C Convention for loading types:
C NTYPE=-2...All body forces (Default for F(T): F(T)=T);
C       =-1...All boundary tractions (Default for F(T): F(T)=T);
C       =>1...As defined by the user for specified nodal loads
C            (concentrated forces) and nonzero displacements
C            (Default for F(T): F(T)=T).
C
C Set default time dependence.
      FVAL=T

C Set time dependence for all body forces.
      IF (NTYPE.EQ.-2)THEN
        FVAL=T

C Set time dependence for all boundary tractions.
      ELSE IF(NTYPE.EQ.-1)THEN
        FVAL=T
      ELSE IF(NTYPE.EQ.1)THEN
        FVAL=T      ! default time dependence
```

```
C Set time dependence for concentrated nodal forces and
C nonzero displacements, as defined below by the user.

C *** The following is for specified nodal displacements
C     at node 2 during rigid-body rotations.
      ELSE IF(NTYPE.EQ.2)THEN
        FVAL=-SIN(T)    ! x displacement component
      ELSE IF(NTYPE.EQ.3)THEN
        FVAL=COS(T)-1. ! y displacement component
C *** End of rigid-body rotation definitions.

      ENDIF
      RETURN
      END
```

The general output file zip2dl.out is listed below.

```
 Large rigid-body rotation test

 NSTART=  0   NSGEN=  1   MODE =  4   IRI= 0   IUPD  =  -1   IAPP = 2
 NPRNT =  1   NGELE=  0   NGNOD=  0   MSD= 0   NODTOT=   0

 TB  = 0.100E+00  TE  = 0.244E+01  DT  = 0.100E+00  TOL = 0.100E-05
 TOLM= 0.100E-03  TOLP= 0.100E-07  TOLY= 0.100E-07  NSUB=  100

 NUMN =     9  NUMEL=     4  NUMAT= 1
 NUMPW=     0  NUMBL=     0  NUMML= 1  NUMSG= 3  MN= 4  NRST= 2  IOPT= 1

 NUMLD=     0  NUMSS=     0  NBODY= 0  NDISP= 1

 NNC  =     0  NMC  =     1  NPD  = 0  NRR  = 1  ACC = 1.00

 MATERIAL/GEOMETRIC SETS
   SET     POIS         WT          THK         YM          SYI     MPAR(1)    MPAR(2)
    1  0.300E+00  0.000E+00  0.100E+01  0.100E+01  0.100E+01     3          0

 Multi-linear material group:
 GLOBAL SET #    LINE SEGMENT #          STRESS         STRAIN
       1               1              0.12000E+01   0.14000E+01
       1               2              0.13000E+01   0.16000E+01
       1               3              0.50000E+00 (Ratio of Modulus E_t/E)

 MATERIAL SET NUMBERS AND CONNECTIVITY VECTORS
       ELE    SET       NODE NUMBERS
         1      1        1      5      9      8
         2      1        5      2      6      9
         3      1        9      6      3      7
         4      1        8      9      7      4


 NODAL COORDINATES
       NODE          X              Y
         1      0.10000E+01    0.10000E+01
         2      0.00000E+00    0.10000E+01
         3      0.00000E+00    0.00000E+00
         4      0.10000E+01    0.00000E+00
         5      0.50000E+00    0.10000E+01
         6      0.00000E+00    0.50000E+00
         7      0.50000E+00    0.00000E+00
         8      0.10000E+01    0.50000E+00
         9      0.50000E+00    0.50000E+00
```

NODAL DISPL	NODE	DOF	TYPE
0.10000E+01	2	1	2
0.00000E+00	3	1	0
0.10000E+01	2	2	3
0.00000E+00	3	2	0

EQUATION NUMBERS

NODE	DOF 1	DOF 2
1	1	2
2	0	0
3	0	0
4	3	4
5	5	6
6	7	8
7	9	10
8	11	12
9	13	14

***** ELEMENT COMPUTATIONS START HERE *****

REQUIRED STORAGE FOR THE STIFFNESS MATRIX = 89

TOTAL REQUIRED STORAGE = 1236 AVAILABLE STORAGE = 200000

FOR ELEMENT 1: AREA = 0.25000E+00
FOR ELEMENT 2: AREA = 0.25000E+00
FOR ELEMENT 3: AREA = 0.25000E+00
FOR ELEMENT 4: AREA = 0.25000E+00

____SOLUTION AT T = 0.24430E+01 IS CONVERGED____
Re-start files now only contain data for this time step;

Number of iteration for this time step is 4;
Total number of iterations so far is 96;

[K] is updated 4 times in this time step;
[K] is updated a total of 96 times so far.

2ND P-K STRESSES & G-L NORMAL STRAINS:

ELE #	COORD X	COORD Y	STRESS X	STRESS Y	STRESS XY	STRAIN X	STRAIN Y
1	0.894E+00	0.894E+00	0.241E-06	0.132E-06	-.681E-07	0.201E-06	0.599E-07
1	0.606E+00	0.894E+00	0.245E-06	0.145E-06	-.124E-06	0.201E-06	0.721E-07
1	0.606E+00	0.606E+00	0.858E-07	0.979E-07	-.119E-06	0.565E-07	0.721E-07
1	0.894E+00	0.606E+00	0.818E-07	0.845E-07	-.634E-07	0.565E-07	0.599E-07
2	0.394E+00	0.894E+00	0.532E-06	0.201E-06	0.155E-07	0.471E-06	0.417E-07
2	0.106E+00	0.894E+00	0.500E-06	0.967E-07	-.108E-06	0.471E-06	-.534E-07
2	0.106E+00	0.606E+00	0.148E-06	-.912E-08	-.145E-06	0.150E-06	-.534E-07
2	0.394E+00	0.606E+00	0.179E-06	0.955E-07	-.211E-07	0.150E-06	0.417E-07
3	0.394E+00	0.394E+00	0.184E-07	0.161E-06	0.565E-07	-.298E-07	0.155E-06
3	0.106E+00	0.394E+00	0.223E-08	0.107E-06	-.960E-08	-.298E-07	0.106E-06
3	0.106E+00	0.106E+00	-.187E-06	0.503E-07	-.285E-07	-.202E-06	0.106E-06
3	0.394E+00	0.106E+00	-.170E-06	0.104E-06	0.376E-07	-.202E-06	0.155E-06
4	0.894E+00	0.394E+00	0.246E-07	0.125E-06	-.313E-07	-.128E-07	0.117E-06
4	0.606E+00	0.394E+00	0.381E-07	0.170E-06	-.486E-07	-.128E-07	0.158E-06
4	0.606E+00	0.106E+00	-.112E-07	0.155E-06	-.329E-07	-.577E-07	0.158E-06
4	0.894E+00	0.106E+00	-.247E-07	0.110E-06	-.156E-07	-.577E-07	0.117E-06

DISPLACEMENTS:

NODE#	COORD X	COORD Y	DISPL X	DISPL Y	DISPL R	DISPL T
1	0.1000E+01	0.1000E+01	-0.2409E+01	-0.1123E+01	-0.2497E+01	0.9095E+00
2	0.0000E+00	0.1000E+01	-0.6431E+00	-0.1766E+01	-0.1766E+01	0.6431E+00
3	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00		
4	0.1000E+01	0.0000E+00	-0.1766E+01	0.6431E+00	-0.1766E+01	0.6431E+00
5	0.5000E+00	0.1000E+01	-0.1526E+01	-0.1444E+01	-0.1974E+01	0.7191E+00

```
  6   0.0000E+00  0.5000E+00 -0.3216E+00 -0.8829E+00 -0.8829E+00  0.3216E+00
  7   0.5000E+00  0.0000E+00 -0.8829E+00  0.3216E+00 -0.8829E+00  0.3216E+00
  8   0.1000E+01  0.5000E+00 -0.2087E+01 -0.2397E+00 -0.1974E+01  0.7191E+00
  9   0.5000E+00  0.5000E+00 -0.1204E+01 -0.5613E+00 -0.1249E+01  0.4548E+00
```

The detailed stresss and strain output file zip2dl.dat is listed below.

```
****** ELEMENT NUMBER     1  ******

***Second Piola-Kirchhoff Stresses:
POINT         X COOR       Y COOR     STRESS X     STRESS Y     STRESS XY
  1      0.89434E+00  0.89434E+00  0.24063E-06  0.13210E-06 -0.68092E-07

Cauchy Stresses:
            STRESS X     STRESS Y    STRESS XY
         0.12867E-06  0.24406E-06 -0.65212E-07

First Piola-Kirchhoff Stresses:
            STRESS X     STRESS Y    STRESS XY    STRESS YX
        -0.14047E-06 -0.14495E-06  0.20690E-06 -0.32815E-07

Principal Cauchy Stresses:
          PSTRESS 1    PSTRESS 2        ANGLE       PSHEAR
         0.27343E-06  0.99292E-07 -0.65750E+02  0.87071E-07

Green-Lagrange Strains:
           STRAIN  X    STRAIN  Y    STRAIN XY  STRS/STRN Z
         0.20100E-06  0.59908E-07 -0.17704E-06 -0.11182E-06

Cauchy Stresses and Green-Lagrange Strains in Polar Coord System:
             R COOR       T COOR     STRESS R     STRESS T    STRESS RT
         0.12648E+01  0.45000E+02  0.12115E-06  0.25157E-06  0.57694E-07

           STRAIN  R    STRAIN  T    STRAIN RT
         0.41934E-07  0.21897E-06 -0.14109E-06

EFFECTIVE STRESS = 0.23974E-06  YIELD SURFACE = 0.10000E+01
EFFECTIVE PLASTIC STRAIN = 0.00000E+00


***Second Piola-Kirchhoff Stresses:
POINT         X COOR       Y COOR     STRESS X     STRESS Y     STRESS XY
  2      0.60566E+00  0.89434E+00  0.24465E-06  0.14550E-06 -0.12368E-06

Cauchy Stresses:
            STRESS X     STRESS Y    STRESS XY
         0.81818E-07  0.30833E-06 -0.70195E-07

First Piola-Kirchhoff Stresses:
            STRESS X     STRESS Y    STRESS XY    STRESS YX
        -0.10780E-06 -0.19095E-06  0.25205E-06  0.11312E-08

Principal Cauchy Stresses:
          PSTRESS 1    PSTRESS 2        ANGLE       PSHEAR
         0.32832E-06  0.61828E-07 -0.74105E+02  0.13324E-06

Green-Lagrange Strains:
           STRAIN  X    STRAIN  Y    STRAIN XY  STRS/STRN Z
         0.20100E-06  0.72101E-07 -0.32156E-06 -0.11704E-06

Cauchy Stresses and Green-Lagrange Strains in Polar Coord System:
             R COOR       T COOR     STRESS R     STRESS T    STRESS RT
         0.10801E+01  0.55893E+02  0.17193E-06  0.21822E-06  0.13122E-06

           STRAIN  R    STRAIN  T    STRAIN RT
```

```
        -0.36666E-07  0.30977E-06 -0.34270E-09

EFFECTIVE STRESS = 0.30218E-06  YIELD SURFACE = 0.10000E+01
EFFECTIVE PLASTIC STRAIN = 0.00000E+00


***Second Piola-Kirchhoff Stresses:
POINT         X COOR       Y COOR      STRESS X     STRESS Y     STRESS XY
  3      0.60566E+00  0.60566E+00  0.85834E-07  0.97851E-07 -0.11899E-06

Cauchy Stresses:
            STRESS X     STRESS Y     STRESS XY
        -0.26394E-07  0.21008E-06 -0.14635E-07

First Piola-Kirchhoff Stresses:
            STRESS X     STRESS Y     STRESS XY    STRESS YX
         0.10799E-07 -0.15146E-06  0.14632E-06  0.28182E-07

Principal Cauchy Stresses:
           PSTRESS 1    PSTRESS 2        ANGLE       PSHEAR
         0.21098E-06 -0.27297E-07 -0.86472E+02  0.11914E-06

Green-Lagrange Strains:
           STRAIN  X    STRAIN  Y    STRAIN XY  STRS/STRN Z
         0.56478E-07  0.72101E-07 -0.30937E-06 -0.55105E-07

Cauchy Stresses and Green-Lagrange Strains in Polar Coord System:
             R COOR       T COOR     STRESS R     STRESS T    STRESS RT
         0.85654E+00  0.45000E+02  0.77207E-07  0.10648E-06  0.11824E-06

           STRAIN  R    STRAIN  T    STRAIN RT
        -0.90394E-07  0.21897E-06  0.15623E-07

EFFECTIVE STRESS = 0.22587E-06  YIELD SURFACE = 0.10000E+01
EFFECTIVE PLASTIC STRAIN = 0.00000E+00


***Second Piola-Kirchhoff Stresses:
POINT         X COOR       Y COOR      STRESS X     STRESS Y     STRESS XY
  4      0.89434E+00  0.60566E+00  0.81814E-07  0.84452E-07 -0.63402E-07

Cauchy Stresses:
            STRESS X     STRESS Y     STRESS XY
         0.20456E-07  0.14581E-06 -0.96527E-08

First Piola-Kirchhoff Stresses:
            STRESS X     STRESS Y     STRESS XY    STRESS YX
        -0.21872E-07 -0.10545E-06  0.10117E-06 -0.57645E-08

Principal Cauchy Stresses:
           PSTRESS 1    PSTRESS 2        ANGLE       PSHEAR
         0.14655E-06  0.19717E-07 -0.85622E+02  0.63416E-07

Green-Lagrange Strains:
           STRAIN  X    STRAIN  Y    STRAIN XY  STRS/STRN Z
         0.56478E-07  0.59908E-07 -0.16485E-06 -0.49880E-07

Cauchy Stresses and Green-Lagrange Strains in Polar Coord System:
             R COOR       T COOR     STRESS R     STRESS T    STRESS RT
         0.10801E+01  0.34107E+02  0.50907E-07  0.11536E-06  0.54617E-07

           STRAIN  R    STRAIN  T    STRAIN RT
        -0.18979E-07  0.13537E-06 -0.57998E-07

EFFECTIVE STRESS = 0.13775E-06  YIELD SURFACE = 0.10000E+01
EFFECTIVE PLASTIC STRAIN = 0.00000E+00
```

****** ELEMENT NUMBER 2 ******

***Second Piola-Kirchhoff Stresses:

POINT	X COOR	Y COOR	STRESS X	STRESS Y	STRESS XY
1	0.39434E+00	0.89434E+00	0.53169E-06	0.20124E-06	0.15482E-07

Cauchy Stresses:

STRESS X	STRESS Y	STRESS XY
0.41026E-06	0.32268E-06	-0.16007E-06

First Piola-Kirchhoff Stresses:

STRESS X	STRESS Y	STRESS XY	STRESS YX
-0.41710E-06	-0.14414E-06	0.33010E-06	-0.14128E-06

Principal Cauchy Stresses:

PSTRESS 1	PSTRESS 2	ANGLE	PSHEAR
0.53242E-06	0.20051E-06	-0.37350E+02	0.16595E-06

Green-Lagrange Strains:

STRAIN X	STRAIN Y	STRAIN XY	STRS/STRN Z
0.47132E-06	0.41730E-07	0.40252E-07	-0.21988E-06

Cauchy Stresses and Green-Lagrange Strains in Polar Coord System:

R COOR	T COOR	STRESS R	STRESS T	STRESS RT
0.97742E+00	0.66206E+02	0.21875E-06	0.51418E-06	0.75629E-07

STRAIN R	STRAIN T	STRAIN RT
0.12651E-06	0.38654E-06	-0.34432E-06

EFFECTIVE STRESS = 0.46574E-06 YIELD SURFACE = 0.10000E+01
EFFECTIVE PLASTIC STRAIN = 0.00000E+00

***Second Piola-Kirchhoff Stresses:

POINT	X COOR	Y COOR	STRESS X	STRESS Y	STRESS XY
2	0.10566E+00	0.89434E+00	0.50032E-06	0.96658E-07	-0.10792E-06

Cauchy Stresses:

STRESS X	STRESS Y	STRESS XY
0.22705E-06	0.36993E-06	-0.21744E-06

First Piola-Kirchhoff Stresses:

STRESS X	STRESS Y	STRESS XY	STRESS YX
-0.31371E-06	-0.14343E-06	0.40442E-06	0.20478E-07

Principal Cauchy Stresses:

PSTRESS 1	PSTRESS 2	ANGLE	PSHEAR
0.52736E-06	0.69615E-07	-0.54094E+02	0.22887E-06

Green-Lagrange Strains:

STRAIN X	STRAIN Y	STRAIN XY	STRS/STRN Z
0.47132E-06	-0.53438E-07	-0.28060E-06	-0.17909E-06

Cauchy Stresses and Green-Lagrange Strains in Polar Coord System:

R COOR	T COOR	STRESS R	STRESS T	STRESS RT
0.90056E+00	0.83262E+02	0.31729E-06	0.27969E-06	0.22810E-06

STRAIN R	STRAIN T	STRAIN RT
-0.78910E-07	0.49679E-06	0.15059E-06

EFFECTIVE STRESS = 0.49623E-06 YIELD SURFACE = 0.10000E+01
EFFECTIVE PLASTIC STRAIN = 0.00000E+00

***Second Piola-Kirchhoff Stresses:

POINT	X COOR	Y COOR	STRESS X	STRESS Y	STRESS XY

```
  3      0.10566E+00  0.60566E+00  0.14773E-06 -0.91183E-08 -0.14453E-06

Cauchy Stresses:
           STRESS X     STRESS Y     STRESS XY
        -0.59500E-07  0.19811E-06 -0.10221E-06

First Piola-Kirchhoff Stresses:
           STRESS X     STRESS Y     STRESS XY     STRESS YX
        -0.20175E-07 -0.85969E-07  0.20568E-06  0.11654E-06

Principal Cauchy Stresses:
          PSTRESS 1    PSTRESS 2        ANGLE       PSHEAR
         0.23374E-06 -0.95127E-07 -0.70784E+02  0.16443E-06

Green-Lagrange Strains:
          STRAIN  X    STRAIN  Y    STRAIN XY  STRS/STRN Z
         0.15047E-06 -0.53438E-07 -0.37577E-06 -0.41584E-07

Cauchy Stresses and Green-Lagrange Strains in Polar Coord System:
             R COOR       T COOR     STRESS R     STRESS T     STRESS RT
         0.61481E+00  0.80104E+02  0.15590E-06 -0.17281E-07  0.13979E-06

           STRAIN  R    STRAIN  T    STRAIN RT
        -0.11104E-06  0.20806E-06  0.28453E-06

EFFECTIVE STRESS = 0.29312E-06  YIELD SURFACE = 0.10000E+01
EFFECTIVE PLASTIC STRAIN = 0.00000E+00


***Second Piola-Kirchhoff Stresses:
POINT         X COOR       Y COOR     STRESS X     STRESS Y     STRESS XY
  4      0.39434E+00  0.60566E+00  0.17911E-06  0.95462E-07 -0.21122E-07

Cauchy Stresses:
           STRESS X     STRESS Y     STRESS XY
         0.12370E-06  0.15086E-06 -0.44842E-07

First Piola-Kirchhoff Stresses:
           STRESS X     STRESS Y     STRESS XY     STRESS YX
        -0.12357E-06 -0.86684E-07  0.13136E-06 -0.45222E-07

Principal Cauchy Stresses:
          PSTRESS 1    PSTRESS 2        ANGLE       PSHEAR
         0.18414E-06  0.90431E-07 -0.53424E+02  0.46853E-07

Green-Lagrange Strains:
          STRAIN  X    STRAIN  Y    STRAIN XY  STRS/STRN Z
         0.15047E-06  0.41730E-07 -0.54916E-07 -0.82371E-07

Cauchy Stresses and Green-Lagrange Strains in Polar Coord System:
             R COOR       T COOR     STRESS R     STRESS T     STRESS RT
         0.72272E+00  0.56932E+02  0.10177E-06  0.17280E-06  0.30561E-07

           STRAIN  R    STRAIN  T    STRAIN RT
         0.48992E-07  0.14321E-06 -0.77223E-07

EFFECTIVE STRESS = 0.15948E-06  YIELD SURFACE = 0.10000E+01
EFFECTIVE PLASTIC STRAIN = 0.00000E+00


****** ELEMENT NUMBER      3  ******

***Second Piola-Kirchhoff Stresses:
POINT         X COOR       Y COOR     STRESS X     STRESS Y     STRESS XY
  1      0.39434E+00  0.39434E+00  0.18437E-07  0.16095E-06  0.56474E-07

Cauchy Stresses:
```

```
          STRESS X      STRESS Y     STRESS XY
        0.13301E-06   0.46379E-07   0.79943E-07

First Piola-Kirchhoff Stresses:
          STRESS X      STRESS Y     STRESS XY     STRESS YX
       -0.50439E-07  -0.86929E-07  -0.31387E-07  -0.14676E-06

Principal Cauchy Stresses:
         PSTRESS 1     PSTRESS 2         ANGLE        PSHEAR
        0.18062E-06  -0.12283E-08   0.30775E+02   0.90923E-07

Green-Lagrange Strains:
          STRAIN  X     STRAIN  Y     STRAIN XY  STRS/STRN Z
       -0.29849E-07   0.15542E-06   0.14683E-06  -0.53817E-07

Cauchy Stresses and Green-Lagrange Strains in Polar Coord System:
            R COOR        T COOR      STRESS R      STRESS T     STRESS RT
        0.55768E+00   0.45000E+02   0.16964E-06   0.97526E-08  -0.43316E-07

          STRAIN  R     STRAIN  T     STRAIN RT
        0.13620E-06  -0.10630E-07   0.18527E-06

EFFECTIVE STRESS = 0.18124E-06  YIELD SURFACE = 0.10000E+01
EFFECTIVE PLASTIC STRAIN = 0.00000E+00


***Second Piola-Kirchhoff Stresses:
POINT          X COOR        Y COOR      STRESS X      STRESS Y     STRESS XY
  2       0.10566E+00   0.39434E+00   0.22310E-08   0.10693E-06  -0.95960E-08

Cauchy Stresses:
          STRESS X      STRESS Y     STRESS XY
        0.36088E-07   0.73078E-07   0.49907E-07

First Piola-Kirchhoff Stresses:
          STRESS X      STRESS Y     STRESS XY     STRESS YX
        0.44632E-08  -0.88056E-07   0.87830E-08  -0.61426E-07

Principal Cauchy Stresses:
         PSTRESS 1     PSTRESS 2         ANGLE        PSHEAR
        0.10781E-06   0.13588E-08   0.55167E+02   0.53224E-07

Green-Lagrange Strains:
          STRAIN  X     STRAIN  Y     STRAIN XY  STRS/STRN Z
       -0.29849E-07   0.10626E-06  -0.24950E-07  -0.32750E-07

Cauchy Stresses and Green-Lagrange Strains in Polar Coord System:
            R COOR        T COOR      STRESS R      STRESS T     STRESS RT
        0.40825E+00   0.75000E+02   0.95553E-07   0.13612E-07  -0.33973E-07

          STRAIN  R     STRAIN  T     STRAIN RT
        0.90910E-07  -0.14494E-07   0.89664E-07

EFFECTIVE STRESS = 0.10713E-06  YIELD SURFACE = 0.10000E+01
EFFECTIVE PLASTIC STRAIN = 0.00000E+00


***Second Piola-Kirchhoff Stresses:
POINT          X COOR        Y COOR      STRESS X      STRESS Y     STRESS XY
  3       0.10566E+00   0.10566E+00  -0.18654E-06   0.50303E-07  -0.28503E-07

Cauchy Stresses:
          STRESS X      STRESS Y     STRESS XY
       -0.11665E-06  -0.19588E-07   0.11172E-06

First Piola-Kirchhoff Stresses:
          STRESS X      STRESS Y     STRESS XY     STRESS YX
```

```
         0.16117E-06 -0.56851E-07 -0.98146E-07 -0.10526E-07

Principal Cauchy Stresses:
           PSTRESS 1     PSTRESS 2         ANGLE        PSHEAR
         0.53685E-07 -0.18992E-06  0.56740E+02  0.12180E-06

Green-Lagrange Strains:
           STRAIN  X     STRAIN  Y     STRAIN XY  STRS/STRN Z
        -0.20163E-06  0.10626E-06 -0.74107E-07  0.40871E-07

Cauchy Stresses and Green-Lagrange Strains in Polar Coord System:
              R COOR        T COOR      STRESS R      STRESS T     STRESS RT
         0.14943E+00  0.45000E+02  0.43599E-07 -0.17984E-06  0.48530E-07

           STRAIN  R     STRAIN  T     STRAIN RT
        -0.84737E-07 -0.10630E-07  0.30790E-06

EFFECTIVE STRESS = 0.22169E-06  YIELD SURFACE = 0.10000E+01
EFFECTIVE PLASTIC STRAIN = 0.00000E+00


***Second Piola-Kirchhoff Stresses:
POINT          X COOR        Y COOR      STRESS X      STRESS Y     STRESS XY
  4       0.39434E+00  0.10566E+00 -0.17033E-06  0.10432E-06  0.37567E-07

Cauchy Stresses:
            STRESS X      STRESS Y     STRESS XY
        -0.19726E-07 -0.46287E-07  0.14175E-06

First Piola-Kirchhoff Stresses:
            STRESS X      STRESS Y     STRESS XY     STRESS YX
         0.10627E-06 -0.55723E-07 -0.13832E-06 -0.95861E-07

Principal Cauchy Stresses:
           PSTRESS 1     PSTRESS 2         ANGLE        PSHEAR
         0.10937E-06 -0.17538E-06  0.42324E+02  0.14237E-06

Green-Lagrange Strains:
           STRAIN  X     STRAIN  Y     STRAIN XY  STRS/STRN Z
        -0.20163E-06  0.15542E-06  0.97675E-07  0.19804E-07

Cauchy Stresses and Green-Lagrange Strains in Polar Coord System:
              R COOR        T COOR      STRESS R      STRESS T     STRESS RT
         0.40825E+00  0.15000E+02  0.49372E-07 -0.11538E-06  0.11612E-06

           STRAIN  R     STRAIN  T     STRAIN RT
        -0.15329E-06  0.10709E-06  0.26312E-06

EFFECTIVE STRESS = 0.24880E-06  YIELD SURFACE = 0.10000E+01
EFFECTIVE PLASTIC STRAIN = 0.00000E+00


****** ELEMENT NUMBER     4  ******

***Second Piola-Kirchhoff Stresses:
POINT          X COOR        Y COOR      STRESS X      STRESS Y     STRESS XY
  1       0.89434E+00  0.39434E+00  0.24611E-07  0.12487E-06 -0.31342E-07

Cauchy Stresses:
            STRESS X      STRESS Y     STRESS XY
         0.35209E-07  0.11427E-06  0.43961E-07

First Piola-Kirchhoff Stresses:
            STRESS X      STRESS Y     STRESS XY     STRESS YX
         0.13116E-08 -0.11578E-06  0.39829E-07 -0.56308E-07

Principal Cauchy Stresses:
```

```
          PSTRESS 1     PSTRESS 2         ANGLE        PSHEAR
        0.13386E-06   0.15619E-07   0.65981E+02   0.59121E-07

Green-Lagrange Strains:
           STRAIN  X     STRAIN  Y     STRAIN XY   STRS/STRN Z
        -0.12849E-07   0.11749E-06 -0.81490E-07 -0.44844E-07

Cauchy Stresses and Green-Lagrange Strains in Polar Coord System:
             R COOR        T COOR      STRESS R      STRESS T     STRESS RT
         0.97742E+00   0.23794E+02   0.80535E-07   0.68944E-07   0.58836E-07

           STRAIN  R     STRAIN  T     STRAIN RT
        -0.21717E-07   0.12635E-06   0.41266E-07

EFFECTIVE STRESS = 0.12677E-06  YIELD SURFACE = 0.10000E+01
EFFECTIVE PLASTIC STRAIN = 0.00000E+00


***Second Piola-Kirchhoff Stresses:
POINT          X COOR        Y COOR      STRESS X      STRESS Y     STRESS XY
  2       0.60566E+00   0.39434E+00   0.38109E-07   0.16986E-06 -0.48606E-07

Cauchy Stresses:
             STRESS X      STRESS Y     STRESS XY
         0.44731E-07   0.16324E-06   0.56490E-07

First Piola-Kirchhoff Stresses:
             STRESS X      STRESS Y     STRESS XY     STRESS YX
         0.20785E-08 -0.16133E-06   0.61729E-07 -0.72025E-07

Principal Cauchy Stresses:
          PSTRESS 1     PSTRESS 2         ANGLE        PSHEAR
        0.18585E-06   0.22118E-07   0.68184E+02   0.81867E-07

Green-Lagrange Strains:
           STRAIN  X     STRAIN  Y     STRAIN XY   STRS/STRN Z
        -0.12849E-07   0.15843E-06 -0.12638E-06 -0.62391E-07

Cauchy Stresses and Green-Lagrange Strains in Polar Coord System:
             R COOR        T COOR      STRESS R      STRESS T     STRESS RT
         0.72272E+00   0.33068E+02   0.13167E-06   0.76299E-07   0.77043E-07

           STRAIN  R     STRAIN  T     STRAIN RT
        -0.19644E-07   0.16522E-06   0.10551E-06

EFFECTIVE STRESS = 0.17584E-06  YIELD SURFACE = 0.10000E+01
EFFECTIVE PLASTIC STRAIN = 0.00000E+00


***Second Piola-Kirchhoff Stresses:
POINT          X COOR        Y COOR      STRESS X      STRESS Y     STRESS XY
  3       0.60566E+00   0.10566E+00 -0.11215E-07   0.15506E-06 -0.32858E-07

Cauchy Stresses:
             STRESS X      STRESS Y     STRESS XY
         0.25198E-07   0.11865E-06   0.76214E-07

First Piola-Kirchhoff Stresses:
             STRESS X      STRESS Y     STRESS XY     STRESS YX
         0.29721E-07 -0.13987E-06   0.17948E-07 -0.74567E-07

Principal Cauchy Stresses:
          PSTRESS 1     PSTRESS 2         ANGLE        PSHEAR
        0.16132E-06 -0.17473E-07   0.60756E+02   0.89398E-07

Green-Lagrange Strains:
           STRAIN  X     STRAIN  Y     STRAIN XY   STRS/STRN Z
```

```
        -0.57735E-07  0.15843E-06 -0.85432E-07 -0.43155E-07

 Cauchy Stresses and Green-Lagrange Strains in Polar Coord System:
              R COOR       T COOR     STRESS R     STRESS T    STRESS RT
          0.61481E+00  0.98961E+01  0.53766E-07  0.90083E-07  0.87534E-07

             STRAIN  R    STRAIN  T    STRAIN RT
         -0.65814E-07  0.16651E-06 -0.71900E-08

 EFFECTIVE STRESS = 0.17073E-06  YIELD SURFACE = 0.10000E+01
 EFFECTIVE PLASTIC STRAIN = 0.00000E+00


 ***Second Piola-Kirchhoff Stresses:
 POINT         X COOR       Y COOR     STRESS X     STRESS Y    STRESS XY
   4       0.89434E+00  0.10566E+00 -0.24713E-07  0.11007E-06 -0.15595E-07

 Cauchy Stresses:
              STRESS X     STRESS Y    STRESS XY
           0.15677E-07  0.69681E-07  0.63685E-07

 First Piola-Kirchhoff Stresses:
              STRESS X     STRESS Y    STRESS XY    STRESS YX
           0.28954E-07 -0.94317E-07 -0.39524E-08 -0.58850E-07

 Principal Cauchy Stresses:
             PSTRESS 1    PSTRESS 2        ANGLE       PSHEAR
           0.11185E-06 -0.26494E-07  0.56488E+02  0.69173E-07

 Green-Lagrange Strains:
             STRAIN  X    STRAIN  Y    STRAIN XY  STRS/STRN Z
          -0.57735E-07  0.11749E-06 -0.40547E-07 -0.25607E-07

 Cauchy Stresses and Green-Lagrange Strains in Polar Coord System:
              R COOR       T COOR     STRESS R     STRESS T    STRESS RT
          0.90056E+00  0.67380E+01  0.31262E-07  0.54096E-07  0.68224E-07

             STRAIN  R    STRAIN  T    STRAIN RT
          -0.60047E-07  0.11980E-06  0.14030E-08

 EFFECTIVE STRESS = 0.12719E-06  YIELD SURFACE = 0.10000E+01
 EFFECTIVE PLASTIC STRAIN = 0.00000E+00
```

REMARKS:

(1) Stresses and strains for rigid-body rotations must be exact zero. The caculculated stresses and strains are on the order of 1.0E-7, which is negligibly small compared to the normalized yield stress of unity and can be considered zero. The calculated values can be reduced further if a smaller iteration tolerance, TOL, is used (TOL=1.0E-6 is used in this example).

(2) Because the stress and strain values are basically round-off errors, which are dicated by the tolerance TOL, the predicted "principal" direction, ANGLE, has no pratical significance.

The plastic-zone output file zip2dl.pls is empty because there is no yielding. The run-time message output file zip2dl.msg is listed below.

```
THE BEGINNING OF A NEW TIME STEP: T =  0.10000E+00.
    Relative errors for force and energy increments: 0.100E+01 & 0.100E+01.
    Relative errors for force and energy increments: 0.357E-01 & 0.138E-01.
    Relative errors for force and energy increments: 0.244E-05 & 0.114E-05.
    Relative errors for force and energy increments: 0.335E-13 & 0.348E-13.

____SOLUTION AT T =  0.10000E+00 IS CONVERGED____
    Re-start files now only contain data for this time step;

    Number of iteration for this time step is    4;
    Total number of iterations so far is      4;

    [K] is updated    4 times in this time step;
    [K] is updated a total of      4 times so far.

    Max. Effective Stress =  0.23674E-07 at Gauss Pt 2 of Element     2
    Max. Relative Eff Strs = 0.236735E-07 at Gauss Pt 2 of Element     2
    (Maximum when divided by initial yield stress)
    (Elastic elements excluded)

    Reactions at nodes with specified displacements:
    NODE#       X FORCE       Y FORCE
        2  -0.67383E-08   0.10345E-08
        3   0.11690E-08  -0.13720E-08

THE BEGINNING OF A NEW TIME STEP: T =  0.20000E+00.
    Relative errors for force and energy increments: 0.100E+01 & 0.100E+01.
    Relative errors for force and energy increments: 0.357E-01 & 0.138E-01.
    Relative errors for force and energy increments: 0.244E-05 & 0.114E-05.
    Relative errors for force and energy increments: 0.335E-13 & 0.348E-13.

____SOLUTION AT T =  0.20000E+00 IS CONVERGED____
    Re-start files now only contain data for this time step;

    Number of iteration for this time step is    4;
    Total number of iterations so far is      8;

    [K] is updated    4 times in this time step;
    [K] is updated a total of      8 times so far.

    Max. Effective Stress =  0.23674E-07 at Gauss Pt 2 of Element     2
    Max. Relative Eff Strs = 0.236735E-07 at Gauss Pt 2 of Element     2
    (Maximum when divided by initial yield stress)
    (Elastic elements excluded)

    Reactions at nodes with specified displacements:
    NODE#       X FORCE       Y FORCE
        2  -0.68079E-08   0.35663E-09
        3   0.13001E-08  -0.12485E-08

THE BEGINNING OF A NEW TIME STEP: T =  0.30000E+00.
    Relative errors for force and energy increments: 0.100E+01 & 0.100E+01.
    Relative errors for force and energy increments: 0.357E-01 & 0.138E-01.
    Relative errors for force and energy increments: 0.244E-05 & 0.114E-05.
    Relative errors for force and energy increments: 0.335E-13 & 0.348E-13.

____SOLUTION AT T =  0.30000E+00 IS CONVERGED____
    Re-start files now only contain data for this time step;

    Number of iteration for this time step is    4;
    Total number of iterations so far is     12;

    [K] is updated    4 times in this time step;
    [K] is updated a total of     12 times so far.

    Max. Effective Stress =  0.23674E-07 at Gauss Pt 2 of Element     2
```

```
    Max. Relative Eff Strs = 0.236735E-07 at Gauss Pt 2 of Element     2
    (Maximum when divided by initial yield stress)
    (Elastic elements excluded)

    Reactions at nodes with specified displacements:
    NODE#       X FORCE       Y FORCE
        2  -0.68095E-08  -0.32481E-09
        3   0.14182E-08  -0.11124E-08

THE BEGINNING OF A NEW TIME STEP: T =  0.40000E+00.
    Relative errors for force and energy increments: 0.100E+01 & 0.100E+01.
    Relative errors for force and energy increments: 0.357E-01 & 0.138E-01.
    Relative errors for force and energy increments: 0.244E-05 & 0.114E-05.
    Relative errors for force and energy increments: 0.335E-13 & 0.348E-13.

____SOLUTION AT T =  0.40000E+00 IS CONVERGED____
    Re-start files now only contain data for this time step;

    Number of iteration for this time step is    4;
    Total number of iterations so far is     16;

    [K] is updated    4 times in this time step;
    [K] is updated a total of     16 times so far.

    Max. Effective Stress =  0.23674E-07 at Gauss Pt 2 of Element     2
    Max. Relative Eff Strs = 0.236735E-07 at Gauss Pt 2 of Element     2
    (Maximum when divided by initial yield stress)
    (Elastic elements excluded)

    Reactions at nodes with specified displacements:
    NODE#       X FORCE       Y FORCE
        2  -0.67431E-08  -0.10030E-08
        3   0.15222E-08  -0.96528E-09

THE BEGINNING OF A NEW TIME STEP: T =  0.50000E+00.
    Relative errors for force and energy increments: 0.100E+01 & 0.100E+01.
    Relative errors for force and energy increments: 0.357E-01 & 0.138E-01.
    Relative errors for force and energy increments: 0.244E-05 & 0.114E-05.
    Relative errors for force and energy increments: 0.335E-13 & 0.348E-13.

____SOLUTION AT T =  0.50000E+00 IS CONVERGED____
    Re-start files now only contain data for this time step;

    Number of iteration for this time step is    4;
    Total number of iterations so far is     20;

    [K] is updated    4 times in this time step;
    [K] is updated a total of     20 times so far.

    Max. Effective Stress =  0.23674E-07 at Gauss Pt 2 of Element     2
    Max. Relative Eff Strs = 0.236735E-07 at Gauss Pt 2 of Element     2
    (Maximum when divided by initial yield stress)
    (Elastic elements excluded)

    Reactions at nodes with specified displacements:
    NODE#       X FORCE       Y FORCE
        2  -0.66093E-08  -0.16712E-08
        3   0.16110E-08  -0.80849E-09

THE BEGINNING OF A NEW TIME STEP: T =  0.60000E+00.
    Relative errors for force and energy increments: 0.100E+01 & 0.100E+01.
    Relative errors for force and energy increments: 0.357E-01 & 0.138E-01.
    Relative errors for force and energy increments: 0.244E-05 & 0.114E-05.
    Relative errors for force and energy increments: 0.335E-13 & 0.348E-13.

____SOLUTION AT T =  0.60000E+00 IS CONVERGED____
    Re-start files now only contain data for this time step;
```

```
    Number of iteration for this time step is     4;
    Total number of iterations so far is      24;

    [K] is updated     4 times in this time step;
    [K] is updated a total of      24 times so far.

    Max. Effective Stress =  0.23674E-07 at Gauss Pt 2 of Element      2
    Max. Relative Eff Strs = 0.236735E-07 at Gauss Pt 2 of Element      2
    (Maximum when divided by initial yield stress)
    (Elastic elements excluded)

    Reactions at nodes with specified displacements:
    NODE#        X FORCE        Y FORCE
        2  -0.64094E-08  -0.23227E-08
        3   0.16836E-08  -0.64362E-09

THE BEGINNING OF A NEW TIME STEP: T =  0.70000E+00.
    Relative errors for force and energy increments: 0.100E+01 & 0.100E+01.
    Relative errors for force and energy increments: 0.357E-01 & 0.138E-01.
    Relative errors for force and energy increments: 0.244E-05 & 0.114E-05.
    Relative errors for force and energy increments: 0.335E-13 & 0.348E-13.

____SOLUTION AT T =  0.70000E+00 IS CONVERGED____
    Re-start files now only contain data for this time step;

    Number of iteration for this time step is     4;
    Total number of iterations so far is      28;

    [K] is updated     4 times in this time step;
    [K] is updated a total of      28 times so far.

    Max. Effective Stress =  0.23674E-07 at Gauss Pt 2 of Element      2
    Max. Relative Eff Strs = 0.236735E-07 at Gauss Pt 2 of Element      2
    (Maximum when divided by initial yield stress)
    (Elastic elements excluded)

    Reactions at nodes with specified displacements:
    NODE#        X FORCE        Y FORCE
        2  -0.61455E-08  -0.29509E-08
        3   0.17395E-08  -0.47232E-09

THE BEGINNING OF A NEW TIME STEP: T =  0.80000E+00.
    Relative errors for force and energy increments: 0.100E+01 & 0.100E+01.
    Relative errors for force and energy increments: 0.357E-01 & 0.138E-01.
    Relative errors for force and energy increments: 0.244E-05 & 0.114E-05.
    Relative errors for force and energy increments: 0.335E-13 & 0.348E-13.

____SOLUTION AT T =  0.80000E+00 IS CONVERGED____
    Re-start files now only contain data for this time step;

    Number of iteration for this time step is     4;
    Total number of iterations so far is      32;

    [K] is updated     4 times in this time step;
    [K] is updated a total of      32 times so far.

    Max. Effective Stress =  0.23674E-07 at Gauss Pt 2 of Element      2
    Max. Relative Eff Strs = 0.236735E-07 at Gauss Pt 2 of Element      2
    (Maximum when divided by initial yield stress)
    (Elastic elements excluded)

    Reactions at nodes with specified displacements:
    NODE#        X FORCE        Y FORCE
        2  -0.58202E-08  -0.35497E-08
        3   0.17779E-08  -0.29631E-09
```

```
THE BEGINNING OF A NEW TIME STEP: T =  0.90000E+00.
    Relative errors for force and energy increments: 0.100E+01 & 0.100E+01.
    Relative errors for force and energy increments: 0.357E-01 & 0.138E-01.
    Relative errors for force and energy increments: 0.244E-05 & 0.114E-05.
    Relative errors for force and energy increments: 0.335E-13 & 0.348E-13.

____SOLUTION AT T =  0.90000E+00 IS CONVERGED____
    Re-start files now only contain data for this time step;

    Number of iteration for this time step is     4;
    Total number of iterations so far is      36;

    [K] is updated     4 times in this time step;
    [K] is updated a total of      36 times so far.

    Max. Effective Stress =  0.23674E-07 at Gauss Pt 2 of Element      2
    Max. Relative Eff Strs = 0.236735E-07 at Gauss Pt 2 of Element      2
    (Maximum when divided by initial yield stress)
    (Elastic elements excluded)

    Reactions at nodes with specified displacements:
    NODE#       X FORCE       Y FORCE
        2  -0.54367E-08  -0.41130E-08
        3   0.17986E-08  -0.11733E-09

THE BEGINNING OF A NEW TIME STEP: T =  0.10000E+01.
    Relative errors for force and energy increments: 0.100E+01 & 0.100E+01.
    Relative errors for force and energy increments: 0.357E-01 & 0.138E-01.
    Relative errors for force and energy increments: 0.244E-05 & 0.114E-05.
    Relative errors for force and energy increments: 0.335E-13 & 0.348E-13.

____SOLUTION AT T =  0.10000E+01 IS CONVERGED____
    Re-start files now only contain data for this time step;

    Number of iteration for this time step is     4;
    Total number of iterations so far is      40;

    [K] is updated     4 times in this time step;
    [K] is updated a total of      40 times so far.

    Max. Effective Stress =  0.23674E-07 at Gauss Pt 2 of Element      2
    Max. Relative Eff Strs = 0.236735E-07 at Gauss Pt 2 of Element      2
    (Maximum when divided by initial yield stress)
    (Elastic elements excluded)

    Reactions at nodes with specified displacements:
    NODE#       X FORCE       Y FORCE
        2  -0.49990E-08  -0.46352E-08
        3   0.18014E-08   0.62820E-10

THE BEGINNING OF A NEW TIME STEP: T =  0.11000E+01.
    Relative errors for force and energy increments: 0.100E+01 & 0.100E+01.
    Relative errors for force and energy increments: 0.357E-01 & 0.138E-01.
    Relative errors for force and energy increments: 0.244E-05 & 0.114E-05.
    Relative errors for force and energy increments: 0.335E-13 & 0.348E-13.

____SOLUTION AT T =  0.11000E+01 IS CONVERGED____
    Re-start files now only contain data for this time step;

    Number of iteration for this time step is     4;
    Total number of iterations so far is      44;

    [K] is updated     4 times in this time step;
    [K] is updated a total of      44 times so far.

    Max. Effective Stress =  0.23674E-07 at Gauss Pt 2 of Element      2
    Max. Relative Eff Strs = 0.236735E-07 at Gauss Pt 2 of Element      2
```

```
     (Maximum when divided by initial yield stress)
     (Elastic elements excluded)

     Reactions at nodes with specified displacements:
     NODE#       X FORCE       Y FORCE
         2  -0.45112E-08  -0.51111E-08
         3   0.17861E-08   0.24234E-09

THE BEGINNING OF A NEW TIME STEP: T =  0.12000E+01.
     Relative errors for force and energy increments: 0.100E+01 & 0.100E+01.
     Relative errors for force and energy increments: 0.357E-01 & 0.138E-01.
     Relative errors for force and energy increments: 0.244E-05 & 0.114E-05.
     Relative errors for force and energy increments: 0.335E-13 & 0.348E-13.

____SOLUTION AT T =  0.12000E+01 IS CONVERGED____
     Re-start files now only contain data for this time step;

     Number of iteration for this time step is    4;
     Total number of iterations so far is     48;

     [K] is updated    4 times in this time step;
     [K] is updated a total of     48 times so far.

     Max. Effective Stress =  0.23674E-07 at Gauss Pt 2 of Element     2
     Max. Relative Eff Strs = 0.236735E-07 at Gauss Pt 2 of Element     2
     (Maximum when divided by initial yield stress)
     (Elastic elements excluded)

     Reactions at nodes with specified displacements:
     NODE#       X FORCE       Y FORCE
         2  -0.39784E-08  -0.55360E-08
         3   0.17530E-08   0.41944E-09

THE BEGINNING OF A NEW TIME STEP: T =  0.13000E+01.
     Relative errors for force and energy increments: 0.100E+01 & 0.100E+01.
     Relative errors for force and energy increments: 0.357E-01 & 0.138E-01.
     Relative errors for force and energy increments: 0.244E-05 & 0.114E-05.
     Relative errors for force and energy increments: 0.335E-13 & 0.348E-13.

____SOLUTION AT T =  0.13000E+01 IS CONVERGED____
     Re-start files now only contain data for this time step;

     Number of iteration for this time step is    4;
     Total number of iterations so far is     52;

     [K] is updated    4 times in this time step;
     [K] is updated a total of     52 times so far.

     Max. Effective Stress =  0.23674E-07 at Gauss Pt 2 of Element     2
     Max. Relative Eff Strs = 0.236735E-07 at Gauss Pt 2 of Element     2
     (Maximum when divided by initial yield stress)
     (Elastic elements excluded)

     Reactions at nodes with specified displacements:
     NODE#       X FORCE       Y FORCE
         2  -0.34059E-08  -0.59055E-08
         3   0.17023E-08   0.59235E-09

THE BEGINNING OF A NEW TIME STEP: T =  0.14000E+01.
     Relative errors for force and energy increments: 0.100E+01 & 0.100E+01.
     Relative errors for force and energy increments: 0.357E-01 & 0.138E-01.
     Relative errors for force and energy increments: 0.244E-05 & 0.114E-05.
     Relative errors for force and energy increments: 0.335E-13 & 0.348E-13.

____SOLUTION AT T =  0.14000E+01 IS CONVERGED____
     Re-start files now only contain data for this time step;
```

```
     Number of iteration for this time step is     4;
     Total number of iterations so far is      56;

     [K] is updated     4 times in this time step;
     [K] is updated a total of      56 times so far.

     Max. Effective Stress =  0.23674E-07 at Gauss Pt 2 of Element      2
     Max. Relative Eff Strs = 0.236735E-07 at Gauss Pt 2 of Element      2
     (Maximum when divided by initial yield stress)
     (Elastic elements excluded)

     Reactions at nodes with specified displacements:
     NODE#        X FORCE        Y FORCE
         2  -0.27993E-08  -0.62160E-08
         3   0.16347E-08   0.75935E-09

THE BEGINNING OF A NEW TIME STEP: T =  0.15000E+01.
     Relative errors for force and energy increments: 0.100E+01 & 0.100E+01.
     Relative errors for force and energy increments: 0.357E-01 & 0.138E-01.
     Relative errors for force and energy increments: 0.244E-05 & 0.114E-05.
     Relative errors for force and energy increments: 0.335E-13 & 0.348E-13.

____SOLUTION AT T =  0.15000E+01 IS CONVERGED____
     Re-start files now only contain data for this time step;

     Number of iteration for this time step is     4;
     Total number of iterations so far is      60;

     [K] is updated     4 times in this time step;
     [K] is updated a total of      60 times so far.

     Max. Effective Stress =  0.23674E-07 at Gauss Pt 2 of Element      2
     Max. Relative Eff Strs = 0.236735E-07 at Gauss Pt 2 of Element      2
     (Maximum when divided by initial yield stress)
     (Elastic elements excluded)

     Reactions at nodes with specified displacements:
     NODE#        X FORCE        Y FORCE
         2  -0.21647E-08  -0.64644E-08
         3   0.15507E-08   0.91875E-09

THE BEGINNING OF A NEW TIME STEP: T =  0.16000E+01.
     Relative errors for force and energy increments: 0.100E+01 & 0.100E+01.
     Relative errors for force and energy increments: 0.357E-01 & 0.138E-01.
     Relative errors for force and energy increments: 0.244E-05 & 0.114E-05.
     Relative errors for force and energy increments: 0.335E-13 & 0.348E-13.

____SOLUTION AT T =  0.16000E+01 IS CONVERGED____
     Re-start files now only contain data for this time step;

     Number of iteration for this time step is     4;
     Total number of iterations so far is      64;

     [K] is updated     4 times in this time step;
     [K] is updated a total of      64 times so far.

     Max. Effective Stress =  0.23674E-07 at Gauss Pt 2 of Element      2
     Max. Relative Eff Strs = 0.236735E-07 at Gauss Pt 2 of Element      2
     (Maximum when divided by initial yield stress)
     (Elastic elements excluded)

     Reactions at nodes with specified displacements:
     NODE#        X FORCE        Y FORCE
         2  -0.15086E-08  -0.66483E-08
         3   0.14513E-08   0.10690E-08

THE BEGINNING OF A NEW TIME STEP: T =  0.17000E+01.
```

```
     Relative errors for force and energy increments: 0.100E+01 & 0.100E+01.
     Relative errors for force and energy increments: 0.357E-01 & 0.138E-01.
     Relative errors for force and energy increments: 0.244E-05 & 0.114E-05.
     Relative errors for force and energy increments: 0.335E-13 & 0.348E-13.

____SOLUTION AT T =  0.17000E+01 IS CONVERGED____
     Re-start files now only contain data for this time step;

     Number of iteration for this time step is     4;
     Total number of iterations so far is      68;

     [K] is updated     4 times in this time step;
     [K] is updated a total of      68 times so far.

     Max. Effective Stress =  0.23674E-07 at Gauss Pt 2 of Element      2
     Max. Relative Eff Strs = 0.236735E-07 at Gauss Pt 2 of Element      2
     (Maximum when divided by initial yield stress)
     (Elastic elements excluded)

     Reactions at nodes with specified displacements:
     NODE#        X FORCE        Y FORCE
         2  -0.83731E-09   -0.67656E-08
         3   0.13373E-08    0.12085E-08

THE BEGINNING OF A NEW TIME STEP: T =  0.18000E+01.
     Relative errors for force and energy increments: 0.100E+01 & 0.100E+01.
     Relative errors for force and energy increments: 0.357E-01 & 0.138E-01.
     Relative errors for force and energy increments: 0.244E-05 & 0.114E-05.
     Relative errors for force and energy increments: 0.335E-13 & 0.348E-13.

____SOLUTION AT T =  0.18000E+01 IS CONVERGED____
     Re-start files now only contain data for this time step;

     Number of iteration for this time step is     4;
     Total number of iterations so far is      72;

     [K] is updated     4 times in this time step;
     [K] is updated a total of      72 times so far.

     Max. Effective Stress =  0.23674E-07 at Gauss Pt 2 of Element      2
     Max. Relative Eff Strs = 0.236735E-07 at Gauss Pt 2 of Element      2
     (Maximum when divided by initial yield stress)
     (Elastic elements excluded)

     Reactions at nodes with specified displacements:
     NODE#        X FORCE        Y FORCE
         2  -0.15769E-09   -0.68154E-08
         3   0.12100E-08    0.13360E-08

THE BEGINNING OF A NEW TIME STEP: T =  0.19000E+01.
     Relative errors for force and energy increments: 0.100E+01 & 0.100E+01.
     Relative errors for force and energy increments: 0.357E-01 & 0.138E-01.
     Relative errors for force and energy increments: 0.244E-05 & 0.114E-05.
     Relative errors for force and energy increments: 0.335E-13 & 0.348E-13.

____SOLUTION AT T =  0.19000E+01 IS CONVERGED____
     Re-start files now only contain data for this time step;

     Number of iteration for this time step is     4;
     Total number of iterations so far is      76;

     [K] is updated     4 times in this time step;
     [K] is updated a total of      76 times so far.

     Max. Effective Stress =  0.23674E-07 at Gauss Pt 2 of Element      2
     Max. Relative Eff Strs = 0.236735E-07 at Gauss Pt 2 of Element      2
     (Maximum when divided by initial yield stress)
```

```
     (Elastic elements excluded)

     Reactions at nodes with specified displacements:
     NODE#        X FORCE        Y FORCE
         2    0.52351E-09  -0.67971E-08
         3    0.10705E-08   0.14501E-08

 THE BEGINNING OF A NEW TIME STEP: T =  0.20000E+01.
     Relative errors for force and energy increments: 0.100E+01 & 0.100E+01.
     Relative errors for force and energy increments: 0.357E-01 & 0.138E-01.
     Relative errors for force and energy increments: 0.244E-05 & 0.114E-05.
     Relative errors for force and energy increments: 0.335E-13 & 0.348E-13.

 ____SOLUTION AT T =  0.20000E+01 IS CONVERGED____
     Re-start files now only contain data for this time step;

     Number of iteration for this time step is     4;
     Total number of iterations so far is      80;

     [K] is updated    4 times in this time step;
     [K] is updated a total of     80 times so far.

     Max. Effective Stress =  0.23674E-07 at Gauss Pt 2 of Element      2
     Max. Relative Eff Strs = 0.236735E-07 at Gauss Pt 2 of Element      2
     (Maximum when divided by initial yield stress)
     (Elastic elements excluded)

     Reactions at nodes with specified displacements:
     NODE#        X FORCE        Y FORCE
         2    0.11995E-08  -0.67109E-08
         3    0.92042E-09   0.15497E-08

 THE BEGINNING OF A NEW TIME STEP: T =  0.21000E+01.
     Relative errors for force and energy increments: 0.100E+01 & 0.100E+01.
     Relative errors for force and energy increments: 0.357E-01 & 0.138E-01.
     Relative errors for force and energy increments: 0.244E-05 & 0.114E-05.
     Relative errors for force and energy increments: 0.335E-13 & 0.348E-13.

 ____SOLUTION AT T =  0.21000E+01 IS CONVERGED____
     Re-start files now only contain data for this time step;

     Number of iteration for this time step is     4;
     Total number of iterations so far is      84;

     [K] is updated    4 times in this time step;
     [K] is updated a total of     84 times so far.

     Max. Effective Stress =  0.23674E-07 at Gauss Pt 2 of Element      2
     Max. Relative Eff Strs = 0.236735E-07 at Gauss Pt 2 of Element      2
     (Maximum when divided by initial yield stress)
     (Elastic elements excluded)

     Reactions at nodes with specified displacements:
     NODE#        X FORCE        Y FORCE
         2    0.18635E-08  -0.65576E-08
         3    0.76111E-09   0.16339E-08

 THE BEGINNING OF A NEW TIME STEP: T =  0.22000E+01.
     Relative errors for force and energy increments: 0.100E+01 & 0.100E+01.
     Relative errors for force and energy increments: 0.357E-01 & 0.138E-01.
     Relative errors for force and energy increments: 0.244E-05 & 0.114E-05.
     Relative errors for force and energy increments: 0.335E-13 & 0.348E-13.

 ____SOLUTION AT T =  0.22000E+01 IS CONVERGED____
     Re-start files now only contain data for this time step;

     Number of iteration for this time step is     4;
```

```
    Total number of iterations so far is     88;

    [K] is updated    4 times in this time step;
    [K] is updated a total of     88 times so far.

    Max. Effective Stress =  0.23674E-07 at Gauss Pt 2 of Element     2
    Max. Relative Eff Strs = 0.236735E-07 at Gauss Pt 2 of Element     2
    (Maximum when divided by initial yield stress)
    (Elastic elements excluded)

    Reactions at nodes with specified displacements:
    NODE#       X FORCE       Y FORCE
        2   0.25088E-08  -0.63388E-08
        3   0.59419E-09   0.17017E-08

THE BEGINNING OF A NEW TIME STEP: T =  0.23000E+01.
    Relative errors for force and energy increments: 0.100E+01 & 0.100E+01.
    Relative errors for force and energy increments: 0.357E-01 & 0.138E-01.
    Relative errors for force and energy increments: 0.244E-05 & 0.114E-05.
    Relative errors for force and energy increments: 0.335E-13 & 0.348E-13.

____SOLUTION AT T =  0.23000E+01 IS CONVERGED____
    Re-start files now only contain data for this time step;

    Number of iteration for this time step is    4;
    Total number of iterations so far is     92;

    [K] is updated    4 times in this time step;
    [K] is updated a total of     92 times so far.

    Max. Effective Stress =  0.23674E-07 at Gauss Pt 2 of Element     2
    Max. Relative Eff Strs = 0.236735E-07 at Gauss Pt 2 of Element     2
    (Maximum when divided by initial yield stress)
    (Elastic elements excluded)

    Reactions at nodes with specified displacements:
    NODE#       X FORCE       Y FORCE
        2   0.31291E-08  -0.60567E-08
        3   0.42133E-09   0.17525E-08

THE BEGINNING OF A NEW TIME STEP: T =  0.24430E+01.
    Relative errors for force and energy increments: 0.100E+01 & 0.100E+01.
    Relative errors for force and energy increments: 0.749E-01 & 0.285E-01.
    Relative errors for force and energy increments: 0.203E-04 & 0.975E-05.
    Relative errors for force and energy increments: 0.564E-11 & 0.721E-11.

____SOLUTION AT T =  0.24430E+01 IS CONVERGED____
    Re-start files now only contain data for this time step;

    Number of iteration for this time step is    4;
    Total number of iterations so far is     96;

    [K] is updated    4 times in this time step;
    [K] is updated a total of     96 times so far.

    Max. Effective Stress =  0.49623E-06 at Gauss Pt 2 of Element     2
    Max. Relative Eff Strs = 0.496231E-06 at Gauss Pt 2 of Element     2
    (Maximum when divided by initial yield stress)
    (Elastic elements excluded)

    Reactions at nodes with specified displacements:
    NODE#       X FORCE       Y FORCE
        2   0.84424E-07  -0.11003E-06
        3  -0.11649E-07   0.41962E-07
```

4.3 CRACK-GROWTH SIMULATION TEST PROBLEMS

Crack-growth simulation calibrations for ZIP2DL have been performed for a variety of cases against results produced by the established code ZIP2D. These test cases cover plane stress, plane strain, and mixed-state of stress problems involving both hypothetical and real elastic-plastic materials. In addition, ZIP2DL has been used with satisfaction in stable crack-growth simulations for the Arcan fracture tests on aluminum alloy specimens conducted at NASA Langley Research Center (see references [9] and [10] in the theoretical manual).

As an example, consider the case of a single-edge crack growing under displacement loading in a sheet made of a hypothetical, linear hardening material, which is the material used in the previous examples with normalized stresses and strains. Due to symmetry, only half of the geometry is modeled, which occupies a 1.6x1.6 domain. An edge crack, with an initial length of 0.5, lies along the left portion of the bottom line.

As shown schematically in Fig. 4.1, the problem domain is first divided uniformly into a 16x16 grid, and then each small square is further divided into to two triangles, leading to a total of 289 nodes and 512 three-noded constant-strain elements. Note that only a few triangles are actually drawn in the figure and only some of the node and element numbers are specified in the schematic. The crack path is identified by the node pairs from node 1 to node 16, with the initial crack tip at node 6. Because of symmetry, the corresponding node numbers for the crack-path node pairs are denoted by -1, as defined by Data Line 21 in the user manual, leading automatically to the satisfaction of symmetry conditions at these nodes. However, since node #17 is not included in the crack path, its symmetry conditions must be specified in zip2dl.in.

A mixed state of stress is assumed, so that all elements, except those in a strip plane strain core, are in plane stress state. The plane strain core strip is located in front of the initial crack tip, where the elements are in plane strain state. This strip region starts at one element behind the initial crack tip and has a height of 0.2 above the crack path. It includes elements 5 through 16, 21 through 32, 37 through 48, and 53 through 54.

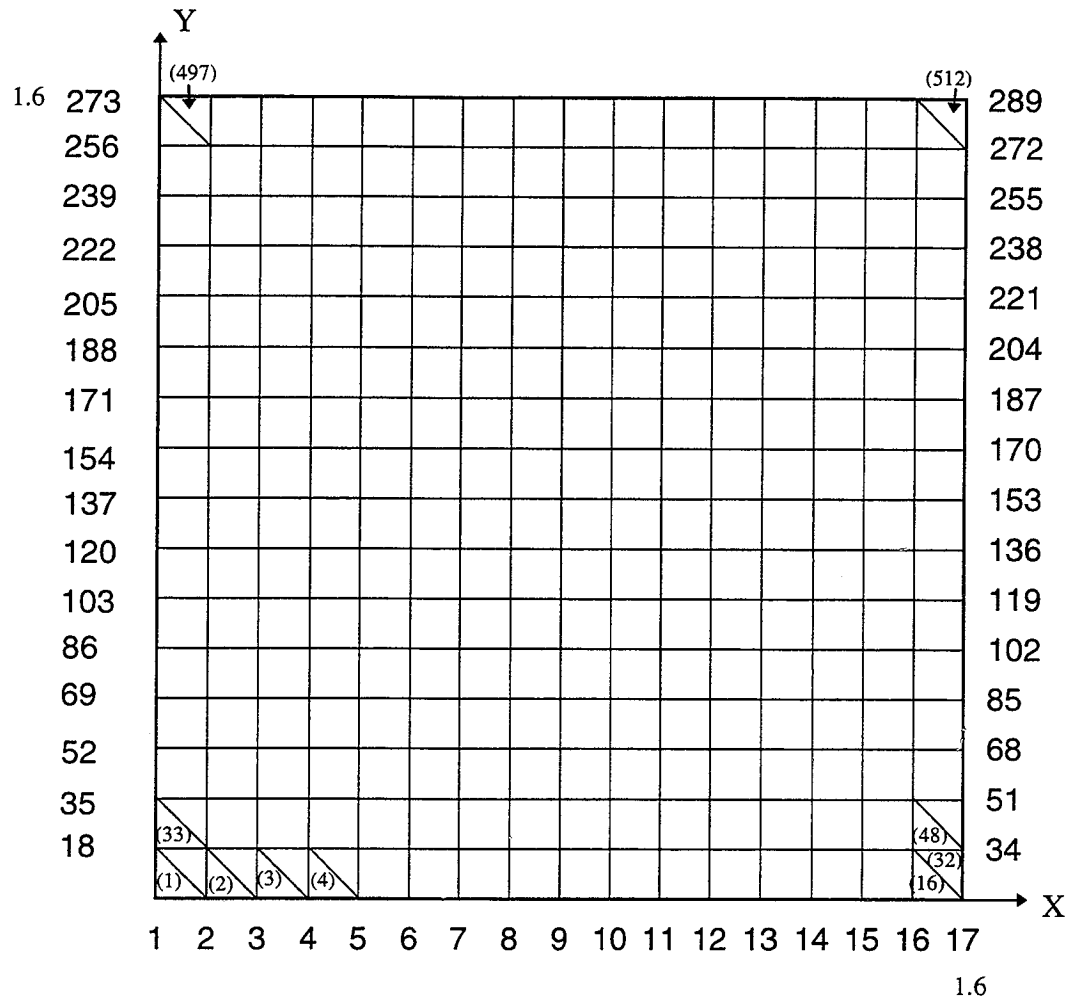


Fig. 4.1 Schematic of the mesh used in the crack growth simulation example

A vertical displacement is specified at node 273, which is at the upper left corner of the domain. This prescribed displacement is increased monotonically from a value that causes initial yielding in the sheet to a value of 10.0 in 100 increments. Rigid-body motion is prevented by restricting the horizontal movement of node 17 at the lower right corner of the domain, along with symmetry conditions along the crack path (which are automatically satisified by setting the node numbers for crack-path nodes on one side of the line to -1; see definition for Data Line 21 in the user manual). To transfer loads to the crack tip effectively and to avoid localized plastic deformation at the point of loading, a strip of elements from the upper boundary of the domain to the line at a distance of 0.2 below it are made to deform only elastically.

A crack-tip opening displacement (CTOD) based fracture criterion is used for the crack-growth simulation. The critical CTOD value is set to 2.0 (note that this is for a hypothetical material) at one node (a distance of 0.1) behind the crack tip. Crack growth occurs when the crack opening displacement at this node is equal to or larger than the critical value at the end of a load step.

The general input file zip2dl.in is listed below.

```
Displ. loading; crack a=0.5; Fake material; Plane strain core
0,1,3,0,0,1,-1,1,1,1,16
0.1318384,1.0, 0.008681616,1.e-6,1.e-4,1.e-8,1.e-8,100
289,512,2,0,1,0,0,3,1,-1
0,0,0,1
0,1,0,1,1.0
1.e+8,1,8,1,-1,3,2.0
1,16,1
1,17,1
.3,0.0,1.0,1.e+0,1.e+0,1
.3,1.0,1.0,1.e+0,1.e+0,-2
1,0.5
17,17,0,0,0
273,273,0,1,0
0,0,0,0,0
273,273,0,0.0,10.0,0,1
0,0,0,0,0,0,0
16
0,6
1,-1
2,-1
3,-1
4,-1
5,-1
6,-1
7,-1
8,-1
9,-1
10,-1
11,-1
12,-1
13,-1
14,-1
15,-1
16,-1
5,16,1
21,32,1
37,48,1
53,64,1
0,0,0
```

The mesh input file zip2dl.msh is listed below.

```
  1, 16,1,1,   2, 18,  1,1,0
 33, 48,1,1,  19, 35, 18,1,0
 65, 80,1,1,  36, 52, 35,1,0
```

```
 97,112,1,1, 53, 69, 52,1,0
129,144,1,1, 70, 86, 69,1,0
161,176,1,1, 87,103, 86,1,0
193,208,1,1,104,120,103,1,0
225,240,1,1,121,137,120,1,0
257,272,1,1,138,154,137,1,0
289,304,1,1,155,171,154,1,0
321,336,1,1,172,188,171,1,0
353,368,1,1,189,205,188,1,0
385,400,1,1,206,222,205,1,0
417,432,1,1,223,239,222,1,0
449,464,1,2,240,256,239,1,0
481,496,1,2,257,273,256,1,0
 17, 32,1,1, 18,  2, 19,1,0
 49, 64,1,1, 35, 19, 36,1,0
 81, 96,1,1, 52, 36, 53,1,0
113,128,1,1, 69, 53, 70,1,0
145,160,1,1, 86, 70, 87,1,0
177,192,1,1,103, 87,104,1,0
209,224,1,1,120,104,121,1,0
241,256,1,1,137,121,138,1,0
273,288,1,1,154,138,155,1,0
305,320,1,1,171,155,172,1,0
337,352,1,1,188,172,189,1,0
369,384,1,1,205,189,206,1,0
401,416,1,1,222,206,223,1,0
433,448,1,1,239,223,240,1,0
465,480,1,2,256,240,257,1,0
497,512,1,2,273,257,274,1,0
   0, 0,0,0,  0,  0,  0,0,0
  1,1,0,0.0,0.0
 17,0,0,1.6,0.0
 18,1,0,0.0,0.1
 34,0,0,1.6,0.1
 35,1,0,0.0,0.2
 51,0,0,1.6,0.2
 52,1,0,0.0,0.3
 68,0,0,1.6,0.3
 69,1,0,0.0,0.4
 85,0,0,1.6,0.4
 86,1,0,0.0,0.5
102,0,0,1.6,0.5
103,1,0,0.0,0.6
119,0,0,1.6,0.6
120,1,0,0.0,0.7
136,0,0,1.6,0.7
137,1,0,0.0,0.8
153,0,0,1.6,0.8
154,1,0,0.0,0.9
170,0,0,1.6,0.9
171,1,0,0.0,1.0
187,0,0,1.6,1,0
188,1,0,0.0,1.1
204,0,0,1.6,1.1
205,1,0,0.0,1.2
221,0,0,1.6,1.2
222,1,0,0.0,1.3
238,0,0,1.6,1.3
239,1,0,0.0,1.4
255,0,0,1.6,1.4
256,1,0,0.0,1.5
272,0,0,1.6,1.5
273,1,0,0.0,1.6
289,0,0,1.6,1.6
  0,0,0,0.0,0.0
```

The general output file zip2dl.out is listed below.

```
Displ. loading; crack a=0.5; Fake material; Plane strain core

NSTART=  0   NSGEN=  1   MODE =  3   IRI= 0   IUPD  =   0   IAPP = 1
NPRNT = -1   NGELE=  1   NGNOD=  1   MSD= 1   NODTOT=  16

TB  = 0.132E+00  TE  = 0.100E+01  DT  = 0.868E-02  TOL = 0.100E-05
TOLM= 0.100E-03  TOLP= 0.100E-07  TOLY= 0.100E-07  NSUB=  100

NUMN =  289  NUMEL=  512  NUMAT= 2
NUMPW=    0  NUMBL=    1  NUMML= 0  NUMSG= 0  MN= 3  NRST= 1  IOPT=-1

NUMLD=    0  NUMSS=    0  NBODY= 0  NDISP= 1

NNC  =    0  NMC  =    1  NPD  = 0  NRR  = 1  ACC = 1.00

ESPR= 0.10000E+09  NGRW=  1  NLMT=  8  NCRI=  1  LCRI= -1  NRLX=  3
CTOD= 0.20000E+01

MATERIAL/GEOMETRIC SETS
  SET     POIS         WT          THK         YM          SYI      MPAR(1)   MPAR(2)
   1  0.300E+00  0.000E+00  0.100E+01  0.100E+01  0.100E+01     1         0
   2  0.300E+00  0.100E+01  0.100E+01  0.100E+01  0.100E+01    -2         0

Bi-linear material group:
GLOBAL SET #      LINE MODULUS RATIO
      1                 0.50000

Number of node pairs for each crack:
   16

Crack tip location for each crack:
    0
    6

CRACK #    PAIR #    Node #1    Node #2
   1          1          1         -1
   1          2          2         -1
   1          3          3         -1
   1          4          4         -1
   1          5          5         -1
   1          6          6         -1
   1          7          7         -1
   1          8          8         -1
   1          9          9         -1
   1         10         10         -1
   1         11         11         -1
   1         12         12         -1
   1         13         13         -1
   1         14         14         -1
   1         15         15         -1
   1         16         16         -1

TOTAL REQUIRED STORAGE FOR CRACK GROWTH DATA =   108;
AVAILABLE STORAGE = 20000


REQUIRED STORAGE FOR THE STIFFNESS MATRIX =   19224

TOTAL REQUIRED STORAGE =   73241   AVAILABLE STORAGE =  200000


*** Node pair at crack tip 2 of crack  1 broke at time  0.65274E+00 ***

Reactions at nodes with specified displacements:
```

```
NODE#        X FORCE        Y FORCE
   17    0.15382E-17    0.85415E-01
  273    0.00000E+00    0.55730E+00

OPENING DISPL OF CRACK  1 JUST BEFORE GROWTH AT TIME  0.65274E+00:
      COD  NODE1     DISPL1X     DISPL1Y  NODE2     DISPL2X     DISPL2Y
0.532E+01      1  -.659E+00  0.266E+01     -1  -.659E+00  -.266E+01
0.464E+01      2  -.662E+00  0.232E+01     -1  -.662E+00  -.232E+01
0.391E+01      3  -.676E+00  0.195E+01     -1  -.676E+00  -.195E+01
0.307E+01      4  -.685E+00  0.153E+01     -1  -.685E+00  -.153E+01
0.202E+01      5  -.660E+00  0.101E+01     -1  -.660E+00  -.101E+01
0.448E-08      6  -.480E+00  0.224E-08     -1  -.480E+00  -.224E-08

___New crack tip information at time  0.65274E+00___
TIP#  CRACK#          DA           X           Y  NODE1  NODE2
   2       1  0.1000E+00  0.6000E+00  0.0000E+00      7     -1

*** Node pair at crack tip 2 of crack  1 broke at time  0.73955E+00 ***

Reactions at nodes with specified displacements:
NODE#        X FORCE        Y FORCE
   17   -0.13102E-16    0.10611E+00
  273    0.00000E+00    0.53785E+00

OPENING DISPL OF CRACK  1 JUST BEFORE GROWTH AT TIME  0.73955E+00:
      COD  NODE1     DISPL1X     DISPL1Y  NODE2     DISPL2X     DISPL2Y
0.741E+01      1  -.820E+00  0.371E+01     -1  -.820E+00  -.371E+01
0.658E+01      2  -.823E+00  0.329E+01     -1  -.823E+00  -.329E+01
0.572E+01      3  -.835E+00  0.286E+01     -1  -.835E+00  -.286E+01
0.477E+01      4  -.850E+00  0.239E+01     -1  -.850E+00  -.239E+01
0.369E+01      5  -.851E+00  0.184E+01     -1  -.851E+00  -.184E+01
0.202E+01      6  -.779E+00  0.101E+01     -1  -.779E+00  -.101E+01
0.515E-08      7  -.628E+00  0.257E-08     -1  -.628E+00  -.257E-08

___New crack tip information at time  0.73955E+00___
TIP#  CRACK#          DA           X           Y  NODE1  NODE2
   2       1  0.2000E+00  0.7000E+00  0.0000E+00      8     -1

*** Node pair at crack tip 2 of crack  1 broke at time  0.76560E+00 ***

Reactions at nodes with specified displacements:
NODE#        X FORCE        Y FORCE
   17   -0.63544E-17    0.12103E+00
  273   -0.43368E-18    0.45836E+00

OPENING DISPL OF CRACK  1 JUST BEFORE GROWTH AT TIME  0.76560E+00:
      COD  NODE1     DISPL1X     DISPL1Y  NODE2     DISPL2X     DISPL2Y
0.892E+01      1  -.875E+00  0.446E+01     -1  -.875E+00  -.446E+01
0.802E+01      2  -.877E+00  0.401E+01     -1  -.877E+00  -.401E+01
0.709E+01      3  -.887E+00  0.355E+01     -1  -.887E+00  -.355E+01
0.611E+01      4  -.900E+00  0.305E+01     -1  -.900E+00  -.305E+01
0.502E+01      5  -.907E+00  0.251E+01     -1  -.907E+00  -.251E+01
0.346E+01      6  -.861E+00  0.173E+01     -1  -.861E+00  -.173E+01
0.200E+01      7  -.840E+00  0.100E+01     -1  -.840E+00  -.100E+01
0.525E-08      8  -.677E+00  0.263E-08     -1  -.677E+00  -.263E-08

___New crack tip information at time  0.76560E+00___
TIP#  CRACK#          DA           X           Y  NODE1  NODE2
   2       1  0.3000E+00  0.8000E+00  0.0000E+00      9     -1

*** Node pair at crack tip 2 of crack  1 broke at time  0.79164E+00 ***

Reactions at nodes with specified displacements:
NODE#        X FORCE        Y FORCE
   17    0.55769E-17    0.13613E+00
  273   -0.43368E-18    0.37706E+00
```

```
OPENING DISPL OF CRACK  1 JUST BEFORE GROWTH AT TIME  0.79164E+00:
      COD  NODE1    DISPL1X    DISPL1Y  NODE2    DISPL2X    DISPL2Y
0.104E+02      1  -.893E+00  0.520E+01     -1  -.893E+00  -.520E+01
0.943E+01      2  -.895E+00  0.471E+01     -1  -.895E+00  -.471E+01
0.844E+01      3  -.903E+00  0.422E+01     -1  -.903E+00  -.422E+01
0.741E+01      4  -.914E+00  0.370E+01     -1  -.914E+00  -.370E+01
0.630E+01      5  -.920E+00  0.315E+01     -1  -.920E+00  -.315E+01
0.474E+01      6  -.879E+00  0.237E+01     -1  -.879E+00  -.237E+01
0.341E+01      7  -.887E+00  0.170E+01     -1  -.887E+00  -.170E+01
0.201E+01      8  -.859E+00  0.100E+01     -1  -.859E+00  -.100E+01
0.528E-08      9  -.686E+00  0.264E-08     -1  -.686E+00  -.264E-08

___New crack tip information at time  0.79164E+00___
TIP#  CRACK#          DA           X           Y  NODE1  NODE2
   2       1  0.4000E+00  0.9000E+00  0.0000E+00     10     -1

*** Node pair at crack tip 2 of crack  1 broke at time  0.82637E+00 ***

Reactions at nodes with specified displacements:
NODE#       X FORCE       Y FORCE
   17  -0.12983E-16   0.15245E+00
  273  -0.43368E-18   0.30109E+00

OPENING DISPL OF CRACK  1 JUST BEFORE GROWTH AT TIME  0.82637E+00:
      COD  NODE1    DISPL1X    DISPL1Y  NODE2    DISPL2X    DISPL2Y
0.120E+02      1  -.885E+00  0.598E+01     -1  -.885E+00  -.598E+01
0.109E+02      2  -.886E+00  0.546E+01     -1  -.886E+00  -.546E+01
0.985E+01      3  -.893E+00  0.493E+01     -1  -.893E+00  -.493E+01
0.876E+01      4  -.901E+00  0.438E+01     -1  -.901E+00  -.438E+01
0.760E+01      5  -.906E+00  0.380E+01     -1  -.906E+00  -.380E+01
0.602E+01      6  -.866E+00  0.301E+01     -1  -.866E+00  -.301E+01
0.470E+01      7  -.881E+00  0.235E+01     -1  -.881E+00  -.235E+01
0.342E+01      8  -.884E+00  0.171E+01     -1  -.884E+00  -.171E+01
0.201E+01      9  -.848E+00  0.101E+01     -1  -.848E+00  -.101E+01
0.526E-08     10  -.667E+00  0.263E-08     -1  -.667E+00  -.263E-08

___New crack tip information at time  0.82637E+00___
TIP#  CRACK#          DA           X           Y  NODE1  NODE2
   2       1  0.5000E+00  0.1000E+01  0.0000E+00     11     -1

*** Node pair at crack tip 2 of crack  1 broke at time  0.86978E+00 ***

Reactions at nodes with specified displacements:
NODE#       X FORCE       Y FORCE
   17  -0.79418E-17   0.16981E+00
  273   0.00000E+00   0.23129E+00

OPENING DISPL OF CRACK  1 JUST BEFORE GROWTH AT TIME  0.86978E+00:
      COD  NODE1    DISPL1X    DISPL1Y  NODE2    DISPL2X    DISPL2Y
0.136E+02      1  -.849E+00  0.680E+01     -1  -.849E+00  -.680E+01
0.125E+02      2  -.850E+00  0.624E+01     -1  -.850E+00  -.624E+01
0.113E+02      3  -.856E+00  0.567E+01     -1  -.856E+00  -.567E+01
0.102E+02      4  -.862E+00  0.509E+01     -1  -.862E+00  -.509E+01
0.895E+01      5  -.865E+00  0.447E+01     -1  -.865E+00  -.447E+01
0.732E+01      6  -.825E+00  0.366E+01     -1  -.825E+00  -.366E+01
0.598E+01      7  -.842E+00  0.299E+01     -1  -.842E+00  -.299E+01
0.471E+01      8  -.855E+00  0.235E+01     -1  -.855E+00  -.235E+01
0.342E+01      9  -.851E+00  0.171E+01     -1  -.851E+00  -.171E+01
0.201E+01     10  -.808E+00  0.100E+01     -1  -.808E+00  -.100E+01
0.515E-08     11  -.623E+00  0.258E-08     -1  -.623E+00  -.258E-08

___New crack tip information at time  0.86978E+00___
TIP#  CRACK#          DA           X           Y  NODE1  NODE2
   2       1  0.6000E+00  0.1100E+01  0.0000E+00     12     -1

*** Node pair at crack tip 2 of crack  1 broke at time  0.93923E+00 ***
```

Reactions at nodes with specified displacements:

NODE#	X FORCE	Y FORCE
17	-0.10625E-16	0.19154E+00
273	0.00000E+00	0.17195E+00

OPENING DISPL OF CRACK 1 JUST BEFORE GROWTH AT TIME 0.93923E+00:

COD	NODE1	DISPL1X	DISPL1Y	NODE2	DISPL2X	DISPL2Y
0.156E+02	1	-.800E+00	0.782E+01	-1	-.800E+00	-.782E+01
0.144E+02	2	-.801E+00	0.720E+01	-1	-.801E+00	-.720E+01
0.132E+02	3	-.806E+00	0.658E+01	-1	-.806E+00	-.658E+01
0.119E+02	4	-.811E+00	0.595E+01	-1	-.811E+00	-.595E+01
0.106E+02	5	-.813E+00	0.529E+01	-1	-.813E+00	-.529E+01
0.886E+01	6	-.772E+00	0.443E+01	-1	-.772E+00	-.443E+01
0.744E+01	7	-.791E+00	0.372E+01	-1	-.791E+00	-.372E+01
0.612E+01	8	-.807E+00	0.306E+01	-1	-.807E+00	-.306E+01
0.481E+01	9	-.814E+00	0.241E+01	-1	-.814E+00	-.241E+01
0.347E+01	10	-.804E+00	0.174E+01	-1	-.804E+00	-.174E+01
0.202E+01	11	-.754E+00	0.101E+01	-1	-.754E+00	-.101E+01
0.502E-08	12	-.562E+00	0.251E-08	-1	-.562E+00	-.251E-08

___New crack tip information at time 0.93923E+00___

TIP#	CRACK#	DA	X	Y	NODE1	NODE2
2	1	0.7000E+00	0.1200E+01	0.0000E+00	13	-1

____SOLUTION AT T = 0.10000E+01 IS CONVERGED____
Re-start files now only contain data for this time step;

Number of iteration for this time step is 5;
Total number of iterations so far is 648;

[K] is updated 0 times in this time step;
[K] is updated a total of 21 times so far.

2ND P-K STRESSES & G-L NORMAL STRAINS:

ELE #	COORD X	COORD Y	STRESS X	STRESS Y	STRESS XY	STRAIN X	STRAIN Y
1	0.333E-01	0.333E-01	-.114E-01	-.114E-01	0.114E-01	-.799E-02	-.799E-02
2	0.133E+00	0.333E-01	-.397E-01	-.422E-02	0.373E-02	-.384E-01	0.770E-02
3	0.233E+00	0.333E-01	-.497E-01	-.105E-03	-.193E-01	-.496E-01	0.148E-01
4	0.333E+00	0.333E-01	-.974E-02	0.932E-02	-.620E-01	-.125E-01	0.122E-01
5	0.433E+00	0.333E-01	-.101E-01	0.863E-01	0.927E-01	0.408E+00	-.279E+00
6	0.533E+00	0.333E-01	0.401E-01	0.284E+00	-.339E-01	-.197E+00	0.573E+00
7	0.633E+00	0.333E-01	0.388E-01	0.360E+00	-.653E-01	-.190E+00	0.656E+00
8	0.733E+00	0.333E-01	0.358E-01	0.388E+00	-.111E+00	-.132E+00	0.627E+00
9	0.833E+00	0.333E-01	0.827E-01	0.388E+00	-.173E+00	-.374E-01	0.556E+00
10	0.933E+00	0.333E-01	0.246E+00	0.406E+00	-.272E+00	0.149E+00	0.456E+00
11	0.103E+01	0.333E-01	0.704E+00	0.528E+00	-.503E+00	0.567E+00	0.326E+00
12	0.113E+01	0.333E-01	0.243E+01	0.123E+01	-.155E+01	0.188E+01	0.259E+00
13	0.123E+01	0.333E-01	0.259E+01	0.400E+01	0.761E+00	0.793E+00	0.281E+01
14	0.133E+01	0.333E-01	0.171E+01	0.147E+01	0.784E+00	0.123E+01	0.457E+00
15	0.143E+01	0.333E-01	0.836E+00	-.245E+00	0.654E+00	0.123E+01	-.939E+00
16	0.153E+01	0.333E-01	0.904E-01	-.167E+01	0.571E+00	0.139E+01	-.227E+01

DISPLACEMENTS:

NODE#	COORD X	COORD Y	DISPL X	DISPL Y	DISPL R	DISPL T
1	0.0000E+00	0.0000E+00	-0.6983E+00	0.8727E+01		
2	0.1000E+00	0.0000E+00	-0.6991E+00	0.8068E+01	-0.6991E+00	0.8068E+01
3	0.2000E+00	0.0000E+00	-0.7030E+00	0.7405E+01	-0.7030E+00	0.7405E+01
4	0.3000E+00	0.0000E+00	-0.7079E+00	0.6730E+01	-0.7079E+00	0.6730E+01
5	0.4000E+00	0.0000E+00	-0.7092E+00	0.6034E+01	-0.7092E+00	0.6034E+01
6	0.5000E+00	0.0000E+00	-0.6683E+00	0.5140E+01	-0.6683E+00	0.5140E+01
7	0.6000E+00	0.0000E+00	-0.6880E+00	0.4399E+01	-0.6880E+00	0.4399E+01
8	0.7000E+00	0.0000E+00	-0.7070E+00	0.3712E+01	-0.7070E+00	0.3712E+01
9	0.8000E+00	0.0000E+00	-0.7203E+00	0.3042E+01	-0.7203E+00	0.3042E+01
10	0.9000E+00	0.0000E+00	-0.7240E+00	0.2374E+01	-0.7240E+00	0.2374E+01
11	0.1000E+01	0.0000E+00	-0.7091E+00	0.1691E+01	-0.7091E+00	0.1691E+01
12	0.1100E+01	0.0000E+00	-0.6524E+00	0.9567E+00	-0.6524E+00	0.9567E+00
13	0.1200E+01	0.0000E+00	-0.4644E+00	0.2316E-08	-0.4644E+00	0.2316E-08

```
   14   0.1300E+01  0.0000E+00 -0.3851E+00  0.5322E-09 -0.3851E+00  0.5322E-09
   15   0.1400E+01  0.0000E+00 -0.2619E+00 -0.2788E-09 -0.2619E+00 -0.2788E-09
   16   0.1500E+01  0.0000E+00 -0.1391E+00 -0.9281E-09 -0.1391E+00 -0.9281E-09
   17   0.1600E+01  0.0000E+00  0.0000E+00  0.0000E+00  0.0000E+00  0.0000E+00
```

The detailed stress and strain output file zip2dl.dat is listed below.

```
****** ELEMENT NUMBER     1  ******

***Cauchy Stresses:
POINT         X COOR       Y COOR     STRESS X     STRESS Y     STRESS XY
  1      0.33333E-01  0.33333E-01 -0.11420E-01 -0.11420E-01  0.11420E-01

Principal Cauchy Stresses:
           PSTRESS 1    PSTRESS 2        ANGLE       PSHEAR
        -0.43368E-18 -0.22841E-01  0.45000E+02  0.11420E-01

Green-Lagrange Strains:
           STRAIN  X    STRAIN  Y    STRAIN XY  STRS/STRN Z
        -0.79942E-02 -0.79942E-02  0.29693E-01  0.68522E-02

Cauchy Stresses and Green-Lagrange Strains in Polar Coord System:
            R COOR       T COOR     STRESS R     STRESS T     STRESS RT
         0.47140E-01  0.45000E+02 -0.65052E-18 -0.22841E-01  0.21684E-18

           STRAIN  R    STRAIN  T    STRAIN RT
         0.68522E-02 -0.22841E-01 -0.80231E-17

EFFECTIVE STRESS = 0.22841E-01  YIELD SURFACE = 0.10000E+01
EFFECTIVE PLASTIC STRAIN = 0.00000E+00


****** ELEMENT NUMBER     2  ******

***Cauchy Stresses:
POINT         X COOR       Y COOR     STRESS X     STRESS Y     STRESS XY
  1      0.13333E+00  0.33333E-01 -0.39704E-01 -0.42161E-02  0.37350E-02

Principal Cauchy Stresses:
           PSTRESS 1    PSTRESS 2        ANGLE       PSHEAR
        -0.38273E-02 -0.40093E-01  0.84057E+02  0.18133E-01

Green-Lagrange Strains:
           STRAIN  X    STRAIN  Y    STRAIN XY  STRS/STRN Z
        -0.38439E-01  0.76951E-02  0.97110E-02  0.13176E-01

Cauchy Stresses and Green-Lagrange Strains in Polar Coord System:
            R COOR       T COOR     STRESS R     STRESS T     STRESS RT
         0.13744E+00  0.14036E+02 -0.35859E-01 -0.80613E-02  0.11646E-01

           STRAIN  R    STRAIN  T    STRAIN RT
        -0.33441E-01  0.26964E-02  0.30279E-01

EFFECTIVE STRESS = 0.38323E-01  YIELD SURFACE = 0.10000E+01
EFFECTIVE PLASTIC STRAIN = 0.00000E+00


****** ELEMENT NUMBER     3  ******

***Cauchy Stresses:
POINT         X COOR       Y COOR     STRESS X     STRESS Y     STRESS XY
  1      0.23333E+00  0.33333E-01 -0.49679E-01 -0.10455E-03 -0.19305E-01

Principal Cauchy Stresses:
           PSTRESS 1    PSTRESS 2        ANGLE       PSHEAR
```

```
         0.65260E-02 -0.56310E-01 -0.71044E+02  0.31418E-01

Green-Lagrange Strains:
           STRAIN  X     STRAIN  Y     STRAIN XY  STRS/STRN Z
        -0.49648E-01  0.14799E-01 -0.50192E-01  0.14935E-01

Cauchy Stresses and Green-Lagrange Strains in Polar Coord System:
             R COOR       T COOR      STRESS R     STRESS T     STRESS RT
         0.23570E+00  0.81301E+01 -0.54093E-01  0.43093E-02 -0.11592E-01

           STRAIN  R     STRAIN  T     STRAIN RT
        -0.55386E-01  0.20537E-01 -0.30139E-01

EFFECTIVE STRESS = 0.59841E-01  YIELD SURFACE = 0.10000E+01
EFFECTIVE PLASTIC STRAIN = 0.00000E+00


****** ELEMENT NUMBER     4  ******

***Cauchy Stresses:
POINT         X COOR       Y COOR      STRESS X     STRESS Y     STRESS XY
  1      0.33333E+00  0.33333E-01 -0.97416E-02  0.93164E-02 -0.62031E-01

Principal Cauchy Stresses:
           PSTRESS 1    PSTRESS 2        ANGLE       PSHEAR
         0.62547E-01 -0.62972E-01 -0.49367E+02  0.62759E-01

Green-Lagrange Strains:
           STRAIN  X     STRAIN  Y     STRAIN XY  STRS/STRN Z
        -0.12537E-01  0.12239E-01 -0.16128E+00  0.12755E-03

Cauchy Stresses and Green-Lagrange Strains in Polar Coord System:
             R COOR       T COOR      STRESS R     STRESS T     STRESS RT
         0.33500E+00  0.57106E+01 -0.21836E-01  0.21411E-01 -0.58916E-01

           STRAIN  R     STRAIN  T     STRAIN RT
        -0.28260E-01  0.27962E-01 -0.15318E+00

EFFECTIVE STRESS = 0.10870E+00  YIELD SURFACE = 0.10000E+01
EFFECTIVE PLASTIC STRAIN = 0.00000E+00


****** ELEMENT NUMBER     5  ******

***Cauchy Stresses:
POINT         X COOR       Y COOR      STRESS X     STRESS Y     STRESS XY
  1      0.43333E+00  0.33333E-01 -0.10083E-01  0.86307E-01  0.92731E-01

Principal Cauchy Stresses:
           PSTRESS 1    PSTRESS 2        ANGLE       PSHEAR
         0.14262E+00 -0.66396E-01  0.58731E+02  0.10451E+00

Green-Lagrange Strains:
           STRAIN  X     STRAIN  Y     STRAIN XY  STRS/STRN Z
         0.40849E+00 -0.27944E+00 -0.19156E+01  0.24640E+00

Cauchy Stresses and Green-Lagrange Strains in Polar Coord System:
             R COOR       T COOR      STRESS R     STRESS T     STRESS RT
         0.43461E+00  0.43987E+01  0.46661E-02  0.71557E-01  0.99011E-01

           STRAIN  R     STRAIN  T     STRAIN RT
         0.25796E+00 -0.12890E+00 -0.19982E+01

EFFECTIVE STRESS = 0.27595E+00  YIELD SURFACE = 0.23497E+01
EFFECTIVE PLASTIC STRAIN = 0.13497E+01
```

****** ELEMENT NUMBER 6 ******

***Cauchy Stresses:

POINT	X COOR	Y COOR	STRESS X	STRESS Y	STRESS XY
1	0.53333E+00	0.33333E-01	0.40147E-01	0.28389E+00	-0.33909E-01

Principal Cauchy Stresses:

PSTRESS 1	PSTRESS 2	ANGLE	PSHEAR
0.28852E+00	0.35518E-01	-0.82226E+02	0.12650E+00

Green-Lagrange Strains:

STRAIN X	STRAIN Y	STRAIN XY	STRS/STRN Z
-0.19693E+00	0.57280E+00	-0.10780E+01	0.61452E+00

Cauchy Stresses and Green-Lagrange Strains in Polar Coord System:

R COOR	T COOR	STRESS R	STRESS T	STRESS RT
0.53437E+00	0.35763E+01	0.36873E-01	0.28716E+00	-0.18470E-01

STRAIN R	STRAIN T	STRAIN RT
-0.26105E+00	0.63691E+00	-0.97372E+00

EFFECTIVE STRESS = 0.50276E+00 YIELD SURFACE = 0.29417E+01
EFFECTIVE PLASTIC STRAIN = 0.19417E+01

****** ELEMENT NUMBER 7 ******

***Cauchy Stresses:

POINT	X COOR	Y COOR	STRESS X	STRESS Y	STRESS XY
1	0.63333E+00	0.33333E-01	0.38806E-01	0.35959E+00	-0.65303E-01

Principal Cauchy Stresses:

PSTRESS 1	PSTRESS 2	ANGLE	PSHEAR
0.37237E+00	0.26022E-01	-0.78923E+02	0.17318E+00

Green-Lagrange Strains:

STRAIN X	STRAIN Y	STRAIN XY	STRS/STRN Z
-0.19001E+00	0.65630E+00	-0.78270E+00	0.76708E+00

Cauchy Stresses and Green-Lagrange Strains in Polar Coord System:

R COOR	T COOR	STRESS R	STRESS T	STRESS RT
0.63421E+00	0.30128E+01	0.32838E-01	0.36556E+00	-0.48105E-01

STRAIN R	STRAIN T	STRAIN RT
-0.22875E+00	0.69505E+00	-0.68954E+00

EFFECTIVE STRESS = 0.64223E+00 YIELD SURFACE = 0.31271E+01
EFFECTIVE PLASTIC STRAIN = 0.21271E+01

****** ELEMENT NUMBER 8 ******

***Cauchy Stresses:

POINT	X COOR	Y COOR	STRESS X	STRESS Y	STRESS XY
1	0.73333E+00	0.33333E-01	0.35802E-01	0.38762E+00	-0.11065E+00

Principal Cauchy Stresses:

PSTRESS 1	PSTRESS 2	ANGLE	PSHEAR
0.41953E+00	0.38971E-02	-0.73915E+02	0.20781E+00

Green-Lagrange Strains:

STRAIN X	STRAIN Y	STRAIN XY	STRS/STRN Z
-0.13239E+00	0.62674E+00	-0.78332E+00	0.81216E+00

Cauchy Stresses and Green-Lagrange Strains in Polar Coord System:

R COOR	T COOR	STRESS R	STRESS T	STRESS RT
0.73409E+00	0.26026E+01	0.26489E-01	0.39693E+00	-0.94231E-01

```
          STRAIN  R     STRAIN  T     STRAIN RT
        -0.16635E+00  0.66070E+00 -0.71122E+00

EFFECTIVE STRESS = 0.70007E+00  YIELD SURFACE = 0.31766E+01
EFFECTIVE PLASTIC STRAIN = 0.21766E+01


****** ELEMENT NUMBER     9  ******

***Cauchy Stresses:
POINT         X COOR        Y COOR      STRESS X      STRESS Y     STRESS XY
  1      0.83333E+00  0.33333E-01  0.82692E-01  0.38761E+00 -0.17261E+00

Principal Cauchy Stresses:
          PSTRESS 1     PSTRESS 2         ANGLE        PSHEAR
        0.46546E+00  0.48507E-02 -0.65726E+02  0.23030E+00

Green-Lagrange Strains:
          STRAIN  X     STRAIN  Y     STRAIN XY  STRS/STRN Z
        -0.37426E-01  0.55643E+00 -0.86904E+00  0.82684E+00

Cauchy Stresses and Green-Lagrange Strains in Polar Coord System:
             R COOR        T COOR      STRESS R      STRESS T     STRESS RT
         0.83400E+00  0.22906E+01  0.69392E-01  0.40091E+00 -0.15988E+00

          STRAIN  R     STRAIN  T     STRAIN RT
        -0.71183E-01  0.59018E+00 -0.81883E+00

EFFECTIVE STRESS = 0.71359E+00  YIELD SURFACE = 0.31588E+01
EFFECTIVE PLASTIC STRAIN = 0.21588E+01


****** ELEMENT NUMBER    10  ******

***Cauchy Stresses:
POINT         X COOR        Y COOR      STRESS X      STRESS Y     STRESS XY
  1      0.93333E+00  0.33333E-01  0.24591E+00  0.40649E+00 -0.27212E+00

Principal Cauchy Stresses:
          PSTRESS 1     PSTRESS 2         ANGLE        PSHEAR
        0.60992E+00  0.42488E-01 -0.53219E+02  0.28372E+00

Green-Lagrange Strains:
          STRAIN  X     STRAIN  Y     STRAIN XY  STRS/STRN Z
        0.14918E+00  0.45603E+00 -0.10421E+01  0.86018E+00

Cauchy Stresses and Green-Lagrange Strains in Polar Coord System:
             R COOR        T COOR      STRESS R      STRESS T     STRESS RT
         0.93393E+00  0.20454E+01  0.22671E+00  0.42570E+00 -0.26570E+00

          STRAIN  R     STRAIN  T     STRAIN RT
        0.11240E+00  0.49281E+00 -0.10175E+01

EFFECTIVE STRESS = 0.72568E+00  YIELD SURFACE = 0.30915E+01
EFFECTIVE PLASTIC STRAIN = 0.20915E+01


****** ELEMENT NUMBER    11  ******

***Cauchy Stresses:
POINT         X COOR        Y COOR      STRESS X      STRESS Y     STRESS XY
  1      0.10333E+01  0.33333E-01  0.70396E+00  0.52844E+00 -0.50335E+00

Principal Cauchy Stresses:
          PSTRESS 1     PSTRESS 2         ANGLE        PSHEAR
        0.11271E+01  0.10525E+00 -0.40055E+02  0.51095E+00
```

```
Green-Lagrange Strains:
          STRAIN  X     STRAIN  Y     STRAIN XY  STRS/STRN Z
        0.56707E+00  0.32630E+00 -0.15551E+01  0.10005E+01

Cauchy Stresses and Green-Lagrange Strains in Polar Coord System:
             R COOR        T COOR      STRESS R      STRESS T     STRESS RT
        0.10339E+01  0.18476E+01  0.67133E+00  0.56106E+00 -0.50796E+00

          STRAIN  R     STRAIN  T     STRAIN RT
        0.51671E+00  0.37667E+00 -0.15674E+01

EFFECTIVE STRESS = 0.96482E+00  YIELD SURFACE = 0.30333E+01
EFFECTIVE PLASTIC STRAIN = 0.20333E+01


****** ELEMENT NUMBER    12  ******

***Cauchy Stresses:
POINT         X COOR        Y COOR      STRESS X      STRESS Y     STRESS XY
  1      0.11333E+01  0.33333E-01  0.24315E+01  0.12301E+01 -0.15467E+01

Principal Cauchy Stresses:
          PSTRESS 1     PSTRESS 2         ANGLE        PSHEAR
        0.34900E+01  0.17157E+00 -0.34387E+02  0.16592E+01

Green-Lagrange Strains:
          STRAIN  X     STRAIN  Y     STRAIN XY  STRS/STRN Z
        0.18797E+01  0.25946E+00 -0.39512E+01  0.16856E+01

Cauchy Stresses and Green-Lagrange Strains in Polar Coord System:
             R COOR        T COOR      STRESS R      STRESS T     STRESS RT
        0.11338E+01  0.16847E+01  0.23395E+01  0.13220E+01 -0.15793E+01

          STRAIN  R     STRAIN  T     STRAIN RT
        0.17622E+01  0.37697E+00 -0.40395E+01

EFFECTIVE STRESS = 0.28775E+01  YIELD SURFACE = 0.28830E+01
EFFECTIVE PLASTIC STRAIN = 0.18830E+01


****** ELEMENT NUMBER    13  ******

***Cauchy Stresses:
POINT         X COOR        Y COOR      STRESS X      STRESS Y     STRESS XY
  1      0.12333E+01  0.33333E-01  0.25927E+01  0.40032E+01  0.76096E+00

Principal Cauchy Stresses:
          PSTRESS 1     PSTRESS 2         ANGLE        PSHEAR
        0.43355E+01  0.22604E+01  0.66412E+02  0.10375E+01

Green-Lagrange Strains:
          STRAIN  X     STRAIN  Y     STRAIN XY  STRS/STRN Z
        0.79342E+00  0.28112E+01  0.33958E+01  0.24147E+01

Cauchy Stresses and Green-Lagrange Strains in Polar Coord System:
             R COOR        T COOR      STRESS R      STRESS T     STRESS RT
        0.12338E+01  0.15482E+01  0.26348E+01  0.39611E+01  0.79794E+00

          STRAIN  R     STRAIN  T     STRAIN RT
        0.88661E+00  0.27180E+01  0.34999E+01

EFFECTIVE STRESS = 0.20024E+01  YIELD SURFACE = 0.20084E+01
EFFECTIVE PLASTIC STRAIN = 0.10084E+01


****** ELEMENT NUMBER    14  ******
```

```
***Cauchy Stresses:
POINT          X COOR        Y COOR      STRESS X     STRESS Y     STRESS XY
  1       0.13333E+01   0.33333E-01   0.17070E+01   0.14706E+01   0.78356E+00

Principal Cauchy Stresses:
           PSTRESS 1     PSTRESS 2         ANGLE        PSHEAR
          0.23813E+01   0.79640E+00   0.40712E+02   0.79242E+00

Green-Lagrange Strains:
            STRAIN  X     STRAIN  Y     STRAIN XY   STRS/STRN Z
          0.12312E+01   0.45728E+00   0.26213E+01   0.10424E+01

Cauchy Stresses and Green-Lagrange Strains in Polar Coord System:
              R COOR        T COOR      STRESS R     STRESS T     STRESS RT
          0.13337E+01   0.14321E+01   0.17460E+01   0.14316E+01   0.77668E+00

            STRAIN  R     STRAIN  T     STRAIN RT
          0.12962E+01   0.39227E+00   0.25794E+01

EFFECTIVE STRESS = 0.14773E+01  YIELD SURFACE = 0.14820E+01
EFFECTIVE PLASTIC STRAIN = 0.48196E+00


****** ELEMENT NUMBER    15  ******

***Cauchy Stresses:
POINT          X COOR        Y COOR      STRESS X     STRESS Y     STRESS XY
  1       0.14333E+01   0.33333E-01   0.83642E+00  -0.24453E+00   0.65353E+00

Principal Cauchy Stresses:
           PSTRESS 1     PSTRESS 2         ANGLE        PSHEAR
          0.11440E+01  -0.55212E+00   0.25204E+02   0.84806E+00

Green-Lagrange Strains:
            STRAIN  X     STRAIN  Y     STRAIN XY   STRS/STRN Z
          0.12279E+01  -0.93949E+00   0.19697E+01   0.12932E+00

Cauchy Stresses and Green-Lagrange Strains in Polar Coord System:
              R COOR        T COOR      STRESS R     STRESS T     STRESS RT
          0.14337E+01   0.13322E+01   0.86621E+00  -0.27433E+00   0.62770E+00

            STRAIN  R     STRAIN  T     STRAIN RT
          0.12726E+01  -0.98410E+00   0.18668E+01

EFFECTIVE STRESS = 0.14783E+01  YIELD SURFACE = 0.14818E+01
EFFECTIVE PLASTIC STRAIN = 0.48181E+00


****** ELEMENT NUMBER    16  ******

***Cauchy Stresses:
POINT          X COOR        Y COOR      STRESS X     STRESS Y     STRESS XY
  1       0.15333E+01   0.33333E-01   0.90355E-01  -0.16707E+01   0.57061E+00

Principal Cauchy Stresses:
            PSTRESS 1     PSTRESS 2         ANGLE        PSHEAR
          0.25908E+00  -0.18394E+01   0.16472E+02   0.10492E+01

Green-Lagrange Strains:
            STRAIN  X     STRAIN  Y     STRAIN XY   STRS/STRN Z
          0.13913E+01  -0.22693E+01   0.18056E+01  -0.61433E+00

Cauchy Stresses and Green-Lagrange Strains in Polar Coord System:
              R COOR        T COOR      STRESS R     STRESS T     STRESS RT
          0.15337E+01   0.12454E+01   0.11432E+00  -0.16946E+01   0.53181E+00
```

```
          STRAIN  R     STRAIN  T     STRAIN RT
        0.14288E+01 -0.23068E+01  0.16448E+01

 EFFECTIVE STRESS = 0.18258E+01  YIELD SURFACE = 0.18293E+01
 EFFECTIVE PLASTIC STRAIN = 0.82934E+00
```

The plastic zone output file zip2dl.pls is listed below.

```
PLASTIC ZONE GIVEN BY YIELDED GAUSS INTEGRATION POINTS:
 (The effective stress is within 2% of the flow stress)

      X-COOR          Y-COOR          EFF-STS         FLOW-STS     PT #      ELE #
 0.11333E+01     0.33333E-01     0.28775E+01     0.28830E+01        1         12
 0.12333E+01     0.33333E-01     0.20024E+01     0.20084E+01        1         13
 0.13333E+01     0.33333E-01     0.14773E+01     0.14820E+01        1         14
 0.14333E+01     0.33333E-01     0.14783E+01     0.14818E+01        1         15
 0.15333E+01     0.33333E-01     0.18258E+01     0.18293E+01        1         16
 0.11667E+01     0.66667E-01     0.20933E+01     0.20991E+01        1         28
 0.15667E+01     0.66667E-01     0.25703E+01     0.25785E+01        1         32
 0.11333E+01     0.13333E+00     0.18000E+01     0.18049E+01        1         44
 0.12333E+01     0.13333E+00     0.17128E+01     0.17170E+01        1         45
 0.13333E+01     0.13333E+00     0.16528E+01     0.16567E+01        1         46
 0.14333E+01     0.13333E+00     0.16959E+01     0.16994E+01        1         47
 0.15333E+01     0.13333E+00     0.20090E+01     0.20139E+01        1         48
 0.11667E+01     0.16667E+00     0.14527E+01     0.14565E+01        1         60
 0.15667E+01     0.16667E+00     0.20849E+01     0.20872E+01        1         64
 0.12333E+01     0.23333E+00     0.19816E+01     0.16578E+01        1         77
 0.13333E+01     0.23333E+00     0.17375E+01     0.16331E+01        1         78
 0.14333E+01     0.23333E+00     0.16438E+01     0.16511E+01        1         79
 0.15333E+01     0.23333E+00     0.19384E+01     0.18846E+01        1         80
 0.11667E+01     0.26667E+00     0.15163E+01     0.12957E+01        1         92
 0.13667E+01     0.26667E+00     0.14082E+01     0.13355E+01        1         94
 0.14667E+01     0.26667E+00     0.22458E+01     0.17110E+01        1         95
 0.15667E+01     0.26667E+00     0.31768E+01     0.22164E+01        1         96
 0.13333E+01     0.33333E+00     0.13137E+01     0.13264E+01        1        110
 0.11667E+01     0.36667E+00     0.10278E+01     0.10307E+01        1        124
 0.14667E+01     0.36667E+00     0.17186E+01     0.14618E+01        1        127
 0.15667E+01     0.36667E+00     0.23217E+01     0.18008E+01        1        128
 0.14667E+01     0.46667E+00     0.12836E+01     0.12639E+01        1        159
 0.15667E+01     0.46667E+00     0.16206E+01     0.14939E+01        1        160
```

The run-time message output file zip2dl.msg is listed below. Because this file is too large, it has been abridged substantially.

```
 THE BEGINNING OF A NEW TIME STEP: T =  0.13184E+00.
     Relative errors for force and energy increments: 0.100E+01 & 0.100E+01.
     Relative errors for force and energy increments: 0.280E-15 & 0.172E-15.

 ____SOLUTION AT T =  0.13184E+00 IS CONVERGED____
     Re-start files now only contain data for this time step;

     Number of iteration for this time step is    2;
     Total number of iterations so far is      2;

     [K] is updated    0 times in this time step;
     [K] is updated a total of      0 times so far.

     Max. Effective Stress =  0.18591E+01 at Gauss Pt 1 of Element   481
     Max. Relative Eff Strs = 0.100000E+01 at Gauss Pt 1 of Element   417
```

```
     (Maximum when divided by initial yield stress)
     (Elastic elements excluded)

     Reactions at nodes with specified displacements:
     NODE#       X FORCE       Y FORCE
        17   0.16427E-14   0.20361E-01
       273   0.27213E-16   0.13116E+00

     Current fracture driving force (CTOD):
     CRACK#    TIP#    CURRENT CTOD    CRITICAL CTOD
          1       2     0.37395E+00      0.20000E+01

THE BEGINNING OF A NEW TIME STEP: T =  0.14052E+00.
     Relative errors for force and energy increments: 0.100E+01 & 0.100E+01.
     Relative errors for force and energy increments: 0.463E-02 & 0.285E-02.
     Relative errors for force and energy increments: 0.923E-04 & 0.563E-04.
     Relative errors for force and energy increments: 0.245E-03 & 0.151E-03.
     Relative errors for force and energy increments: 0.944E-05 & 0.579E-05.
     Relative errors for force and energy increments: 0.510E-31 & 0.204E-31.

____SOLUTION AT T =  0.14052E+00 IS CONVERGED____
     Re-start files now only contain data for this time step;

     Number of iteration for this time step is     6;
     Total number of iterations so far is      8;

     [K] is updated    0 times in this time step;
     [K] is updated a total of      0 times so far.

     Max. Effective Stress =  0.19779E+01 at Gauss Pt 1 of Element   481
     Max. Relative Eff Strs = 0.104502E+01 at Gauss Pt 1 of Element   417
     (Maximum when divided by initial yield stress)
     (Elastic elements excluded)

     Reactions at nodes with specified displacements:
     NODE#       X FORCE       Y FORCE
        17  -0.54210E-19   0.21669E-01
       273   0.00000E+00   0.13958E+00

     Max. and min. subincrement numbers: MCMAX=   328 & MCMIN=      1.

     Current fracture driving force (CTOD):
     CRACK#    TIP#    CURRENT CTOD    CRITICAL CTOD
          1       2     0.39796E+00      0.20000E+01

THE BEGINNING OF A NEW TIME STEP: T =  0.64405E+00.
     Relative errors for force and energy increments: 0.100E+01 & 0.100E+01.
     Relative errors for force and energy increments: 0.195E-01 & 0.233E-01.
     Relative errors for force and energy increments: 0.108E-02 & 0.520E-03.
     Relative errors for force and energy increments: 0.740E-03 & 0.946E-03.
     Relative errors for force and energy increments: 0.112E-04 & 0.629E-05.
     Relative errors for force and energy increments: 0.739E-30 & 0.574E-30.

____SOLUTION AT T =  0.64405E+00 IS CONVERGED____
     Re-start files now only contain data for this time step;

     Number of iteration for this time step is     6;
     Total number of iterations so far is    356;

     [K] is updated    0 times in this time step;
     [K] is updated a total of      0 times so far.

     Max. Effective Stress =  0.77735E+01 at Gauss Pt 1 of Element   481
     Max. Relative Eff Strs = 0.375216E+01 at Gauss Pt 1 of Element   417
     (Maximum when divided by initial yield stress)
     (Elastic elements excluded)
```

```
     Reactions at nodes with specified displacements:
     NODE#       X FORCE       Y FORCE
        17  -0.67424E-17   0.84594E-01
       273   0.00000E+00   0.55134E+00

     Max. and min. subincrement numbers: MCMAX=    298 & MCMIN=      1.

     Current fracture driving force (CTOD):
     CRACK#    TIP#    CURRENT CTOD    CRITICAL CTOD
          1       2     0.19884E+01      0.20000E+01

 THE BEGINNING OF A NEW TIME STEP: T =  0.65274E+00.
     Relative errors for force and energy increments: 0.100E+01 & 0.100E+01.
     Relative errors for force and energy increments: 0.193E-01 & 0.235E-01.
     Relative errors for force and energy increments: 0.112E-02 & 0.527E-03.
     Relative errors for force and energy increments: 0.742E-03 & 0.954E-03.
     Relative errors for force and energy increments: 0.115E-04 & 0.680E-05.
     Relative errors for force and energy increments: 0.585E-30 & 0.438E-30.

 ____SOLUTION AT T =  0.65274E+00 IS CONVERGED____
     Re-start files now only contain data for this time step;

     Number of iteration for this time step is     6;
     Total number of iterations so far is    362;

     [K] is updated    0 times in this time step;
     [K] is updated a total of      0 times so far.

     Max. Effective Stress =  0.78574E+01 at Gauss Pt 1 of Element   481
     Max. Relative Eff Strs = 0.379314E+01 at Gauss Pt 1 of Element   417
     (Maximum when divided by initial yield stress)
     (Elastic elements excluded)

     Reactions at nodes with specified displacements:
     NODE#       X FORCE       Y FORCE
        17   0.15382E-17   0.85415E-01
       273   0.00000E+00   0.55730E+00

     Max. and min. subincrement numbers: MCMAX=    297 & MCMIN=      1.

     Current fracture driving force (CTOD):
     CRACK#    TIP#    CURRENT CTOD    CRITICAL CTOD
          1       2     0.20161E+01      0.20000E+01

 *** SOLUTION DIVERGED AT T = 0.65274E+00 & ISUB =  9.
     Total diverged iterations at this time step is  1.
     Relative errors for force and energy increments: 0.500E+02 & 0.687E+02.

 *** SOLUTION DIVERGED AT T = 0.65274E+00 & ISUB = 10.
     Total diverged iterations at this time step is  2.
     Relative errors for force and energy increments: 0.667E+01 & 0.342E+01.
     Relative errors for force and energy increments: 0.398E+00 & 0.149E+00.
     Relative errors for force and energy increments: 0.261E+00 & 0.152E+00.
     Relative errors for force and energy increments: 0.149E-01 & 0.109E-01.
     Relative errors for force and energy increments: 0.274E-07 & 0.982E-08.

 ____SOLUTION AT T =  0.65274E+00 IS CONVERGED____
     Re-start files now only contain data for this time step;

     Number of iteration for this time step is    14;
     Total number of iterations so far is    370;

     [K] is updated    3 times in this time step;
     [K] is updated a total of      3 times so far.

     Max. Effective Stress =  0.66811E+01 at Gauss Pt 1 of Element   481
     Max. Relative Eff Strs = 0.316119E+01 at Gauss Pt 1 of Element   417
```

```
     (Maximum when divided by initial yield stress)
     (Elastic elements excluded)

     Reactions at nodes with specified displacements:
     NODE#        X FORCE        Y FORCE
        17   -0.21711E-16    0.94464E-01
       273   -0.86736E-18    0.47424E+00

     Max. and min. subincrement numbers: MCMAX=   4733 & MCMIN=      1.

     Current fracture driving force (CTOD):
     CRACK#     TIP#     CURRENT CTOD     CRITICAL CTOD
          1        2      0.16913E+01       0.20000E+01

THE BEGINNING OF A NEW TIME STEP: T =  0.73087E+00.
     Relative errors for force and energy increments: 0.100E+01 & 0.100E+01.
     Relative errors for force and energy increments: 0.114E-01 & 0.149E-01.
     Relative errors for force and energy increments: 0.765E-03 & 0.581E-03.
     Relative errors for force and energy increments: 0.484E-03 & 0.637E-03.
     Relative errors for force and energy increments: 0.753E-04 & 0.682E-04.
     Relative errors for force and energy increments: 0.710E-30 & 0.365E-30.

____SOLUTION AT T =  0.73087E+00 IS CONVERGED____
     Re-start files now only contain data for this time step;

     Number of iteration for this time step is     6;
     Total number of iterations so far is     424;

     [K] is updated     0 times in this time step;
     [K] is updated a total of       3 times so far.

     Max. Effective Stress =  0.74972E+01 at Gauss Pt 1 of Element    481
     Max. Relative Eff Strs = 0.359929E+01 at Gauss Pt 1 of Element    417
     (Maximum when divided by initial yield stress)
     (Elastic elements excluded)

     Reactions at nodes with specified displacements:
     NODE#        X FORCE        Y FORCE
        17   -0.67559E-17    0.10496E+00
       273    0.00000E+00    0.53184E+00

     Max. and min. subincrement numbers: MCMAX=    183 & MCMIN=      1.

     Current fracture driving force (CTOD):
     CRACK#     TIP#     CURRENT CTOD     CRITICAL CTOD
          1        2      0.19871E+01       0.20000E+01

THE BEGINNING OF A NEW TIME STEP: T =  0.73955E+00.
     Relative errors for force and energy increments: 0.100E+01 & 0.100E+01.
     Relative errors for force and energy increments: 0.115E-01 & 0.153E-01.
     Relative errors for force and energy increments: 0.726E-03 & 0.562E-03.
     Relative errors for force and energy increments: 0.507E-03 & 0.666E-03.
     Relative errors for force and energy increments: 0.690E-04 & 0.639E-04.
     Relative errors for force and energy increments: 0.555E-30 & 0.250E-30.

____SOLUTION AT T =  0.73955E+00 IS CONVERGED____
     Re-start files now only contain data for this time step;

     Number of iteration for this time step is     6;
     Total number of iterations so far is     430;

     [K] is updated     0 times in this time step;
     [K] is updated a total of       3 times so far.

     Max. Effective Stress =  0.75823E+01 at Gauss Pt 1 of Element    481
     Max. Relative Eff Strs = 0.364513E+01 at Gauss Pt 1 of Element    417
     (Maximum when divided by initial yield stress)
```

```
     (Elastic elements excluded)

     Reactions at nodes with specified displacements:
     NODE#       X FORCE       Y FORCE
        17  -0.13102E-16   0.10611E+00
       273   0.00000E+00   0.53785E+00

     Max. and min. subincrement numbers: MCMAX=   183 & MCMIN=     1.

     Current fracture driving force (CTOD):
     CRACK#    TIP#    CURRENT CTOD    CRITICAL CTOD
          1       2     0.20233E+01      0.20000E+01

*** SOLUTION DIVERGED AT T = 0.73955E+00 & ISUB =  9.
     Total diverged iterations at this time step is  1.
     Relative errors for force and energy increments: 0.661E+02 & 0.877E+02.

*** SOLUTION DIVERGED AT T = 0.73955E+00 & ISUB = 10.
     Total diverged iterations at this time step is  2.
     Relative errors for force and energy increments: 0.888E+01 & 0.432E+01.
     Relative errors for force and energy increments: 0.507E+00 & 0.180E+00.
     Relative errors for force and energy increments: 0.350E+00 & 0.196E+00.
     Relative errors for force and energy increments: 0.168E-01 & 0.122E-01.
     Relative errors for force and energy increments: 0.576E-30 & 0.396E-30.

____SOLUTION AT T =  0.73955E+00 IS CONVERGED____
     Re-start files now only contain data for this time step;

     Number of iteration for this time step is   14;
     Total number of iterations so far is    438;

     [K] is updated    3 times in this time step;
     [K] is updated a total of      6 times so far.

     Max. Effective Stress =  0.62172E+01 at Gauss Pt 1 of Element   481
     Max. Relative Eff Strs = 0.291343E+01 at Gauss Pt 1 of Element   417
     (Maximum when divided by initial yield stress)
     (Elastic elements excluded)

     Reactions at nodes with specified displacements:
     NODE#       X FORCE       Y FORCE
        17  -0.22208E-16   0.11688E+00
       273   0.00000E+00   0.44148E+00

     Max. and min. subincrement numbers: MCMAX=  5905 & MCMIN=     1.

     Current fracture driving force (CTOD):
     CRACK#    TIP#    CURRENT CTOD    CRITICAL CTOD
          1       2     0.19208E+01      0.20000E+01

THE BEGINNING OF A NEW TIME STEP: T =  0.75691E+00.
     Relative errors for force and energy increments: 0.100E+01 & 0.100E+01.
     Relative errors for force and energy increments: 0.650E-02 & 0.759E-02.
     Relative errors for force and energy increments: 0.311E-03 & 0.205E-03.
     Relative errors for force and energy increments: 0.324E-03 & 0.368E-03.
     Relative errors for force and energy increments: 0.259E-04 & 0.230E-04.
     Relative errors for force and energy increments: 0.223E-08 & 0.557E-09.

____SOLUTION AT T =  0.75691E+00 IS CONVERGED____
     Re-start files now only contain data for this time step;

     Number of iteration for this time step is    6;
     Total number of iterations so far is    450;

     [K] is updated    0 times in this time step;
     [K] is updated a total of      6 times so far.
```

```
     Max. Effective Stress =  0.63817E+01 at Gauss Pt 1 of Element   481
     Max. Relative Eff Strs = 0.300125E+01 at Gauss Pt 1 of Element   417
     (Maximum when divided by initial yield stress)
     (Elastic elements excluded)

     Reactions at nodes with specified displacements:
     NODE#       X FORCE       Y FORCE
        17   0.81654E-18   0.11979E+00
       273   0.00000E+00   0.45309E+00

     Max. and min. subincrement numbers: MCMAX=   134 & MCMIN=     1.

     Current fracture driving force (CTOD):
     CRACK#    TIP#    CURRENT CTOD    CRITICAL CTOD
          1       2     0.19754E+01      0.20000E+01

THE BEGINNING OF A NEW TIME STEP: T =  0.76560E+00.
     Relative errors for force and energy increments: 0.100E+01 & 0.100E+01.
     Relative errors for force and energy increments: 0.791E-02 & 0.990E-02.
     Relative errors for force and energy increments: 0.519E-03 & 0.260E-03.
     Relative errors for force and energy increments: 0.409E-03 & 0.475E-03.
     Relative errors for force and energy increments: 0.348E-04 & 0.257E-04.
     Relative errors for force and energy increments: 0.610E-30 & 0.418E-30.

____SOLUTION AT T =  0.76560E+00 IS CONVERGED____
     Re-start files now only contain data for this time step;

     Number of iteration for this time step is    6;
     Total number of iterations so far is    456;

     [K] is updated    0 times in this time step;
     [K] is updated a total of      6 times so far.

     Max. Effective Stress =  0.64564E+01 at Gauss Pt 1 of Element   481
     Max. Relative Eff Strs = 0.304119E+01 at Gauss Pt 1 of Element   417
     (Maximum when divided by initial yield stress)
     (Elastic elements excluded)

     Reactions at nodes with specified displacements:
     NODE#       X FORCE       Y FORCE
        17  -0.63544E-17   0.12103E+00
       273  -0.43368E-18   0.45836E+00

     Max. and min. subincrement numbers: MCMAX=   131 & MCMIN=     1.

     Current fracture driving force (CTOD):
     CRACK#    TIP#    CURRENT CTOD    CRITICAL CTOD
          1       2     0.20047E+01      0.20000E+01

*** SOLUTION DIVERGED AT T = 0.76560E+00 & ISUB =  9.
     Total diverged iterations at this time step is  1.
     Relative errors for force and energy increments: 0.689E+02 & 0.886E+02.

*** SOLUTION DIVERGED AT T = 0.76560E+00 & ISUB = 10.
     Total diverged iterations at this time step is  2.
     Relative errors for force and energy increments: 0.891E+01 & 0.385E+01.
     Relative errors for force and energy increments: 0.503E+00 & 0.146E+00.
     Relative errors for force and energy increments: 0.354E+00 & 0.182E+00.
     Relative errors for force and energy increments: 0.117E-01 & 0.700E-02.
     Relative errors for force and energy increments: 0.262E-06 & 0.936E-07.

____SOLUTION AT T =  0.76560E+00 IS CONVERGED____
     Re-start files now only contain data for this time step;

     Number of iteration for this time step is   14;
     Total number of iterations so far is    464;
```

```
     [K] is updated    3 times in this time step;
     [K] is updated a total of      9 times so far.

     Max. Effective Stress =  0.51167E+01 at Gauss Pt 1 of Element   481
     Max. Relative Eff Strs = 0.293380E+01 at Gauss Pt 1 of Element     8
     (Maximum when divided by initial yield stress)
     (Elastic elements excluded)

     Reactions at nodes with specified displacements:
     NODE#       X FORCE       Y FORCE
        17  -0.35602E-16   0.13172E+00
       273   0.00000E+00   0.36379E+00

     Max. and min. subincrement numbers: MCMAX=  6067 & MCMIN=      1.

     Current fracture driving force (CTOD):
     CRACK#    TIP#    CURRENT CTOD    CRITICAL CTOD
          1       2     0.19263E+01      0.20000E+01

THE BEGINNING OF A NEW TIME STEP: T =  0.77428E+00.
     Relative errors for force and energy increments: 0.100E+01 & 0.100E+01.
     Relative errors for force and energy increments: 0.256E-02 & 0.156E-02.
     Relative errors for force and energy increments: 0.131E-03 & 0.742E-04.
     Relative errors for force and energy increments: 0.130E-03 & 0.649E-04.
     Relative errors for force and energy increments: 0.143E-04 & 0.100E-04.
     Relative errors for force and energy increments: 0.540E-30 & 0.254E-30.

_____SOLUTION AT T =  0.77428E+00 IS CONVERGED_____
     Re-start files now only contain data for this time step;

     Number of iteration for this time step is    6;
     Total number of iterations so far is    470;

     [K] is updated    0 times in this time step;
     [K] is updated a total of      9 times so far.

     Max. Effective Stress =  0.51849E+01 at Gauss Pt 1 of Element   481
     Max. Relative Eff Strs = 0.297807E+01 at Gauss Pt 1 of Element     8
     (Maximum when divided by initial yield stress)
     (Elastic elements excluded)

     Reactions at nodes with specified displacements:
     NODE#       X FORCE       Y FORCE
        17  -0.50009E-17   0.13344E+00
       273   0.00000E+00   0.36860E+00

     Max. and min. subincrement numbers: MCMAX=   105 & MCMIN=      1.

     Current fracture driving force (CTOD):
     CRACK#    TIP#    CURRENT CTOD    CRITICAL CTOD
          1       2     0.19527E+01      0.20000E+01

THE BEGINNING OF A NEW TIME STEP: T =  0.78296E+00.
     Relative errors for force and energy increments: 0.100E+01 & 0.100E+01.
     Relative errors for force and energy increments: 0.673E-02 & 0.759E-02.
     Relative errors for force and energy increments: 0.331E-03 & 0.205E-03.
     Relative errors for force and energy increments: 0.376E-03 & 0.368E-03.
     Relative errors for force and energy increments: 0.229E-04 & 0.220E-04.
     Relative errors for force and energy increments: 0.674E-30 & 0.261E-30.

_____SOLUTION AT T =  0.78296E+00 IS CONVERGED_____
     Re-start files now only contain data for this time step;

     Number of iteration for this time step is    6;
     Total number of iterations so far is    476;

     [K] is updated    0 times in this time step;
```

```
     [K] is updated a total of      9 times so far.

     Max. Effective Stress =  0.52458E+01 at Gauss Pt 1 of Element   481
     Max. Relative Eff Strs = 0.302275E+01 at Gauss Pt 1 of Element     8
     (Maximum when divided by initial yield stress)
     (Elastic elements excluded)

     Reactions at nodes with specified displacements:
     NODE#       X FORCE       Y FORCE
        17  -0.88430E-17   0.13478E+00
       273   0.00000E+00   0.37290E+00

     Max. and min. subincrement numbers: MCMAX=   134 & MCMIN=     1.

     Current fracture driving force (CTOD):
     CRACK#    TIP#    CURRENT CTOD    CRITICAL CTOD
          1       2     0.19802E+01      0.20000E+01

THE BEGINNING OF A NEW TIME STEP: T =  0.79164E+00.
     Relative errors for force and energy increments: 0.100E+01 & 0.100E+01.
     Relative errors for force and energy increments: 0.710E-02 & 0.855E-02.
     Relative errors for force and energy increments: 0.375E-03 & 0.218E-03.
     Relative errors for force and energy increments: 0.382E-03 & 0.414E-03.
     Relative errors for force and energy increments: 0.204E-04 & 0.197E-04.
     Relative errors for force and energy increments: 0.877E-30 & 0.761E-30.

____SOLUTION AT T =  0.79164E+00 IS CONVERGED____
     Re-start files now only contain data for this time step;

     Number of iteration for this time step is    6;
     Total number of iterations so far is    482;

     [K] is updated    0 times in this time step;
     [K] is updated a total of      9 times so far.

     Max. Effective Stress =  0.53047E+01 at Gauss Pt 1 of Element   481
     Max. Relative Eff Strs = 0.306836E+01 at Gauss Pt 1 of Element     8
     (Maximum when divided by initial yield stress)
     (Elastic elements excluded)

     Reactions at nodes with specified displacements:
     NODE#       X FORCE       Y FORCE
        17   0.55769E-17   0.13613E+00
       273  -0.43368E-18   0.37706E+00

     Max. and min. subincrement numbers: MCMAX=   122 & MCMIN=     1.

     Current fracture driving force (CTOD):
     CRACK#    TIP#    CURRENT CTOD    CRITICAL CTOD
          1       2     0.20085E+01      0.20000E+01

*** SOLUTION DIVERGED AT T = 0.79164E+00 & ISUB =  9.
     Total diverged iterations at this time step is  1.
     Relative errors for force and energy increments: 0.697E+02 & 0.872E+02.

*** SOLUTION DIVERGED AT T = 0.79164E+00 & ISUB = 10.
     Total diverged iterations at this time step is  2.
     Relative errors for force and energy increments: 0.872E+01 & 0.326E+01.
     Relative errors for force and energy increments: 0.494E+00 & 0.122E+00.
     Relative errors for force and energy increments: 0.335E+00 & 0.156E+00.
     Relative errors for force and energy increments: 0.726E-02 & 0.396E-02.
     Relative errors for force and energy increments: 0.429E-30 & 0.227E-30.

____SOLUTION AT T =  0.79164E+00 IS CONVERGED____
     Re-start files now only contain data for this time step;

     Number of iteration for this time step is   14;
```

```
     Total number of iterations so far is    490;

     [K] is updated    3 times in this time step;
     [K] is updated a total of     12 times so far.

     Max. Effective Stress =  0.40399E+01 at Gauss Pt 1 of Element   481
     Max. Relative Eff Strs = 0.292372E+01 at Gauss Pt 1 of Element     9
     (Maximum when divided by initial yield stress)
     (Elastic elements excluded)

     Reactions at nodes with specified displacements:
     NODE#       X FORCE       Y FORCE
        17  -0.13607E-16   0.14625E+00
       273  -0.43368E-18   0.28776E+00

     Max. and min. subincrement numbers: MCMAX=  6060 & MCMIN=     1.

     Current fracture driving force (CTOD):
     CRACK#    TIP#    CURRENT CTOD    CRITICAL CTOD
          1       2     0.19088E+01      0.20000E+01

THE BEGINNING OF A NEW TIME STEP: T =  0.81769E+00.
     Relative errors for force and energy increments: 0.100E+01 & 0.100E+01.
     Relative errors for force and energy increments: 0.561E-02 & 0.645E-02.
     Relative errors for force and energy increments: 0.281E-03 & 0.172E-03.
     Relative errors for force and energy increments: 0.281E-03 & 0.299E-03.
     Relative errors for force and energy increments: 0.126E-04 & 0.143E-04.
     Relative errors for force and energy increments: 0.556E-30 & 0.173E-30.

____SOLUTION AT T =  0.81769E+00 IS CONVERGED____
     Re-start files now only contain data for this time step;

     Number of iteration for this time step is    6;
     Total number of iterations so far is    508;

     [K] is updated    0 times in this time step;
     [K] is updated a total of     12 times so far.

     Max. Effective Stress =  0.41843E+01 at Gauss Pt 1 of Element   481
     Max. Relative Eff Strs = 0.304912E+01 at Gauss Pt 1 of Element     9
     (Maximum when divided by initial yield stress)
     (Elastic elements excluded)

     Reactions at nodes with specified displacements:
     NODE#       X FORCE       Y FORCE
        17  -0.98933E-18   0.15095E+00
       273   0.43368E-18   0.29795E+00

     Max. and min. subincrement numbers: MCMAX=   134 & MCMIN=     1.

     Current fracture driving force (CTOD):
     CRACK#    TIP#    CURRENT CTOD    CRITICAL CTOD
          1       2     0.19872E+01      0.20000E+01

THE BEGINNING OF A NEW TIME STEP: T =  0.82637E+00.
     Relative errors for force and energy increments: 0.100E+01 & 0.100E+01.
     Relative errors for force and energy increments: 0.550E-02 & 0.709E-02.
     Relative errors for force and energy increments: 0.347E-03 & 0.206E-03.
     Relative errors for force and energy increments: 0.281E-03 & 0.323E-03.
     Relative errors for force and energy increments: 0.134E-04 & 0.159E-04.
     Relative errors for force and energy increments: 0.500E-30 & 0.189E-30.

____SOLUTION AT T =  0.82637E+00 IS CONVERGED____
     Re-start files now only contain data for this time step;

     Number of iteration for this time step is    6;
     Total number of iterations so far is    514;
```

```
    [K] is updated    0 times in this time step;
    [K] is updated a total of     12 times so far.

    Max. Effective Stress =  0.42287E+01 at Gauss Pt 1 of Element   481
    Max. Relative Eff Strs = 0.309103E+01 at Gauss Pt 1 of Element     9
    (Maximum when divided by initial yield stress)
    (Elastic elements excluded)

    Reactions at nodes with specified displacements:
    NODE#       X FORCE       Y FORCE
       17  -0.12983E-16   0.15245E+00
      273  -0.43368E-18   0.30109E+00

    Max. and min. subincrement numbers: MCMAX=   126 & MCMIN=     1.

    Current fracture driving force (CTOD):
    CRACK#    TIP#    CURRENT CTOD    CRITICAL CTOD
         1       2     0.20143E+01      0.20000E+01

*** SOLUTION DIVERGED AT T = 0.82637E+00 & ISUB =  9.
    Total diverged iterations at this time step is  1.
    Relative errors for force and energy increments: 0.692E+02 & 0.846E+02.

*** SOLUTION DIVERGED AT T = 0.82637E+00 & ISUB = 10.
    Total diverged iterations at this time step is  2.
    Relative errors for force and energy increments: 0.834E+01 & 0.275E+01.
    Relative errors for force and energy increments: 0.478E+00 & 0.112E+00.
    Relative errors for force and energy increments: 0.315E+00 & 0.131E+00.
    Relative errors for force and energy increments: 0.355E-02 & 0.188E-02.
    Relative errors for force and energy increments: 0.383E-04 & 0.997E-05.
    Relative errors for force and energy increments: 0.144E-09 & 0.921E-10.

____SOLUTION AT T =  0.82637E+00 IS CONVERGED____
    Re-start files now only contain data for this time step;

    Number of iteration for this time step is   15;
    Total number of iterations so far is    523;

    [K] is updated    3 times in this time step;
    [K] is updated a total of     15 times so far.

    Max. Effective Stress =  0.30667E+01 at Gauss Pt 1 of Element   481
    Max. Relative Eff Strs = 0.286850E+01 at Gauss Pt 1 of Element    10
    (Maximum when divided by initial yield stress)
    (Elastic elements excluded)

    Reactions at nodes with specified displacements:
    NODE#       X FORCE       Y FORCE
       17   0.21640E-05   0.16159E+00
      273   0.00000E+00   0.21901E+00

    Max. and min. subincrement numbers: MCMAX=  5903 & MCMIN=     1.

    Current fracture driving force (CTOD):
    CRACK#    TIP#    CURRENT CTOD    CRITICAL CTOD
         1       2     0.18796E+01      0.20000E+01

THE BEGINNING OF A NEW TIME STEP: T =  0.86109E+00.
    Relative errors for force and energy increments: 0.100E+01 & 0.100E+01.
    Relative errors for force and energy increments: 0.465E-02 & 0.438E-02.
    Relative errors for force and energy increments: 0.252E-03 & 0.105E-03.
    Relative errors for force and energy increments: 0.232E-03 & 0.203E-03.
    Relative errors for force and energy increments: 0.784E-05 & 0.526E-05.
    Relative errors for force and energy increments: 0.511E-30 & 0.271E-30.

____SOLUTION AT T =  0.86109E+00 IS CONVERGED____
```

```
    Re-start files now only contain data for this time step;

    Number of iteration for this time step is     6;
    Total number of iterations so far is     547;

    [K] is updated     0 times in this time step;
    [K] is updated a total of      15 times so far.

    Max. Effective Stress =  0.32082E+01 at Gauss Pt 1 of Element    481
    Max. Relative Eff Strs = 0.302168E+01 at Gauss Pt 1 of Element     10
    (Maximum when divided by initial yield stress)
    (Elastic elements excluded)

    Reactions at nodes with specified displacements:
    NODE#        X FORCE        Y FORCE
       17    0.21413E-17    0.16819E+00
      273    0.00000E+00    0.22901E+00

    Max. and min. subincrement numbers: MCMAX=    129 & MCMIN=      1.

    Current fracture driving force (CTOD):
    CRACK#     TIP#     CURRENT CTOD     CRITICAL CTOD
         1        2      0.19785E+01       0.20000E+01

THE BEGINNING OF A NEW TIME STEP: T =  0.86978E+00.
    Relative errors for force and energy increments: 0.100E+01 & 0.100E+01.
    Relative errors for force and energy increments: 0.621E-02 & 0.538E-02.
    Relative errors for force and energy increments: 0.301E-03 & 0.128E-03.
    Relative errors for force and energy increments: 0.324E-03 & 0.255E-03.
    Relative errors for force and energy increments: 0.124E-04 & 0.795E-05.
    Relative errors for force and energy increments: 0.481E-30 & 0.175E-30.

____SOLUTION AT T =  0.86978E+00 IS CONVERGED____
    Re-start files now only contain data for this time step;

    Number of iteration for this time step is     6;
    Total number of iterations so far is     553;

    [K] is updated     0 times in this time step;
    [K] is updated a total of      15 times so far.

    Max. Effective Stress =  0.32404E+01 at Gauss Pt 1 of Element    481
    Max. Relative Eff Strs = 0.306357E+01 at Gauss Pt 1 of Element     10
    (Maximum when divided by initial yield stress)
    (Elastic elements excluded)

    Reactions at nodes with specified displacements:
    NODE#        X FORCE        Y FORCE
       17   -0.79418E-17    0.16981E+00
      273    0.00000E+00    0.23129E+00

    Max. and min. subincrement numbers: MCMAX=    129 & MCMIN=      1.

    Current fracture driving force (CTOD):
    CRACK#     TIP#     CURRENT CTOD     CRITICAL CTOD
         1        2      0.20053E+01       0.20000E+01

*** SOLUTION DIVERGED AT T = 0.86978E+00 & ISUB =  9.
    Total diverged iterations at this time step is  1.
    Relative errors for force and energy increments: 0.663E+02 & 0.797E+02.

*** SOLUTION DIVERGED AT T = 0.86978E+00 & ISUB = 10.
    Total diverged iterations at this time step is  2.
    Relative errors for force and energy increments: 0.739E+01 & 0.217E+01.
    Relative errors for force and energy increments: 0.469E+00 & 0.103E+00.
    Relative errors for force and energy increments: 0.260E+00 & 0.101E+00.
    Relative errors for force and energy increments: 0.224E-02 & 0.117E-02.
```

```
     Relative errors for force and energy increments: 0.269E-05 & 0.796E-06.
     Relative errors for force and energy increments: 0.326E-30 & 0.176E-30.

_____SOLUTION AT T =  0.86978E+00 IS CONVERGED_____
     Re-start files now only contain data for this time step;

     Number of iteration for this time step is    15;
     Total number of iterations so far is     562;

     [K] is updated     3 times in this time step;
     [K] is updated a total of      18 times so far.

     Max. Effective Stress =  0.27770E+01 at Gauss Pt 1 of Element     11
     Max. Relative Eff Strs = 0.277698E+01 at Gauss Pt 1 of Element     11
     (Maximum when divided by initial yield stress)
     (Elastic elements excluded)

     Reactions at nodes with specified displacements:
     NODE#        X FORCE        Y FORCE
        17  -0.30358E-17    0.17762E+00
       273   0.00000E+00    0.15857E+00

     Max. and min. subincrement numbers: MCMAX=  5434 & MCMIN=      1.

     Current fracture driving force (CTOD):
     CRACK#     TIP#     CURRENT CTOD     CRITICAL CTOD
          1        2      0.18224E+01       0.20000E+01

THE BEGINNING OF A NEW TIME STEP: T =  0.93055E+00.
     Relative errors for force and energy increments: 0.100E+01 & 0.100E+01.
     Relative errors for force and energy increments: 0.704E-02 & 0.463E-02.
     Relative errors for force and energy increments: 0.216E-03 & 0.770E-04.
     Relative errors for force and energy increments: 0.345E-03 & 0.225E-03.
     Relative errors for force and energy increments: 0.438E-05 & 0.228E-05.
     Relative errors for force and energy increments: 0.385E-30 & 0.202E-30.

_____SOLUTION AT T =  0.93055E+00 IS CONVERGED_____
     Re-start files now only contain data for this time step;

     Number of iteration for this time step is     6;
     Total number of iterations so far is     603;

     [K] is updated     0 times in this time step;
     [K] is updated a total of      18 times so far.

     Max. Effective Stress =  0.30062E+01 at Gauss Pt 1 of Element     11
     Max. Relative Eff Strs = 0.300620E+01 at Gauss Pt 1 of Element     11
     (Maximum when divided by initial yield stress)
     (Elastic elements excluded)

     Reactions at nodes with specified displacements:
     NODE#        X FORCE        Y FORCE
        17  -0.75894E-18    0.18991E+00
       273   0.00000E+00    0.17046E+00

     Max. and min. subincrement numbers: MCMAX=   153 & MCMIN=      1.

     Current fracture driving force (CTOD):
     CRACK#     TIP#     CURRENT CTOD     CRITICAL CTOD
          1        2      0.19899E+01       0.20000E+01

THE BEGINNING OF A NEW TIME STEP: T =  0.93923E+00.
     Relative errors for force and energy increments: 0.100E+01 & 0.100E+01.
     Relative errors for force and energy increments: 0.665E-02 & 0.448E-02.
     Relative errors for force and energy increments: 0.216E-03 & 0.767E-04.
     Relative errors for force and energy increments: 0.323E-03 & 0.215E-03.
     Relative errors for force and energy increments: 0.465E-05 & 0.242E-05.
```

```
    Relative errors for force and energy increments: 0.347E-30 & 0.524E-30.

____SOLUTION AT T =  0.93923E+00 IS CONVERGED____
    Re-start files now only contain data for this time step;

    Number of iteration for this time step is    6;
    Total number of iterations so far is    609;

    [K] is updated    0 times in this time step;
    [K] is updated a total of     18 times so far.

    Max. Effective Stress =  0.30288E+01 at Gauss Pt 1 of Element    11
    Max. Relative Eff Strs = 0.302880E+01 at Gauss Pt 1 of Element    11
    (Maximum when divided by initial yield stress)
    (Elastic elements excluded)

    Reactions at nodes with specified displacements:
    NODE#       X FORCE       Y FORCE
       17  -0.10625E-16   0.19154E+00
      273   0.00000E+00   0.17195E+00

    Max. and min. subincrement numbers: MCMAX=   145 & MCMIN=     1.

    Current fracture driving force (CTOD):
    CRACK#    TIP#    CURRENT CTOD    CRITICAL CTOD
         1       2     0.20168E+01      0.20000E+01

*** SOLUTION DIVERGED AT T = 0.93923E+00 & ISUB =  9.
    Total diverged iterations at this time step is  1.
    Relative errors for force and energy increments: 0.628E+02 & 0.750E+02.

*** SOLUTION DIVERGED AT T = 0.93923E+00 & ISUB = 10.
    Total diverged iterations at this time step is  2.
    Relative errors for force and energy increments: 0.560E+01 & 0.156E+01.
    Relative errors for force and energy increments: 0.353E+00 & 0.706E-01.
    Relative errors for force and energy increments: 0.193E+00 & 0.751E-01.
    Relative errors for force and energy increments: 0.493E-03 & 0.280E-03.
    Relative errors for force and energy increments: 0.247E-05 & 0.802E-06.
    Relative errors for force and energy increments: 0.327E-07 & 0.937E-08.

____SOLUTION AT T =  0.93923E+00 IS CONVERGED____
    Re-start files now only contain data for this time step;

    Number of iteration for this time step is   15;
    Total number of iterations so far is    618;

    [K] is updated    3 times in this time step;
    [K] is updated a total of     21 times so far.

    Max. Effective Stress =  0.26789E+01 at Gauss Pt 1 of Element    12
    Max. Relative Eff Strs = 0.267891E+01 at Gauss Pt 1 of Element    12
    (Maximum when divided by initial yield stress)
    (Elastic elements excluded)

    Reactions at nodes with specified displacements:
    NODE#       X FORCE       Y FORCE
       17  -0.53126E-17   0.19703E+00
      273  -0.21684E-18   0.10927E+00

    Max. and min. subincrement numbers: MCMAX=  6207 & MCMIN=     1.

    Current fracture driving force (CTOD):
    CRACK#    TIP#    CURRENT CTOD    CRITICAL CTOD
         1       2     0.17630E+01      0.20000E+01

THE BEGINNING OF A NEW TIME STEP: T =  0.98264E+00.
    Relative errors for force and energy increments: 0.100E+01 & 0.100E+01.
```

```
     Relative errors for force and energy increments: 0.359E-02 & 0.175E-02.
     Relative errors for force and energy increments: 0.144E-03 & 0.399E-04.
     Relative errors for force and energy increments: 0.146E-03 & 0.741E-04.
     Relative errors for force and energy increments: 0.649E-06 & 0.303E-06.

 ____SOLUTION AT T =  0.98264E+00 IS CONVERGED____
     Re-start files now only contain data for this time step;

     Number of iteration for this time step is     5;
     Total number of iterations so far is    643;

     [K] is updated    0 times in this time step;
     [K] is updated a total of     21 times so far.

     Max. Effective Stress =  0.28292E+01 at Gauss Pt 1 of Element    12
     Max. Relative Eff Strs = 0.282917E+01 at Gauss Pt 1 of Element    12
     (Maximum when divided by initial yield stress)
     (Elastic elements excluded)

     Reactions at nodes with specified displacements:
     NODE#        X FORCE        Y FORCE
        17  -0.10946E-05    0.20756E+00
       273   0.00000E+00    0.11515E+00

     Max. and min. subincrement numbers: MCMAX=   133 & MCMIN=     6.

     Current fracture driving force (CTOD):
     CRACK#     TIP#     CURRENT CTOD     CRITICAL CTOD
          1        2      0.18667E+01       0.20000E+01

 THE BEGINNING OF A NEW TIME STEP: T =  0.10000E+01.
     Relative errors for force and energy increments: 0.100E+01 & 0.100E+01.
     Relative errors for force and energy increments: 0.428E-02 & 0.226E-02.
     Relative errors for force and energy increments: 0.130E-03 & 0.469E-04.
     Relative errors for force and energy increments: 0.192E-03 & 0.103E-03.
     Relative errors for force and energy increments: 0.767E-06 & 0.404E-06.

 ____SOLUTION AT T =  0.10000E+01 IS CONVERGED____
     Re-start files now only contain data for this time step;

     Number of iteration for this time step is     5;
     Total number of iterations so far is    648;

     [K] is updated    0 times in this time step;
     [K] is updated a total of     21 times so far.

     Max. Effective Stress =  0.28775E+01 at Gauss Pt 1 of Element    12
     Max. Relative Eff Strs = 0.287753E+01 at Gauss Pt 1 of Element    12
     (Maximum when divided by initial yield stress)
     (Elastic elements excluded)

     Reactions at nodes with specified displacements:
     NODE#        X FORCE        Y FORCE
        17  -0.27409E-05    0.21111E+00
       273   0.00000E+00    0.11719E+00

     Max. and min. subincrement numbers: MCMAX=   279 & MCMIN=     1.

     Current fracture driving force (CTOD):
     CRACK#     TIP#     CURRENT CTOD     CRITICAL CTOD
          1        2      0.19134E+01       0.20000E+01
```

REPORT DOCUMENTATION PAGE

Form Approved
OMB No. 0704-0188

1. AGENCY USE ONLY (*Leave blank*)	2. REPORT DATE	3. REPORT TYPE AND DATES COVERED
	April 1997	Technical Memorandum

4. TITLE AND SUBTITLE
ZIP2DL: An Elastic-Plastic, Large-Rotation Finite-Element Stress Analysis and Crack-Growth Simulation Program

5. FUNDING NUMBERS
538-02-10-01

6. AUTHOR(S)
Xiaomin Deng
James C. Newman, Jr.

7. PERFORMING ORGANIZATION NAME(S) AND ADDRESS(ES)
NASA Langley Research Center
Hampton, Virginia 23681-0001

8. PERFORMING ORGANIZATION REPORT NUMBER

9. SPONSORING / MONITORING AGENCY NAME(S) AND ADDRESS(ES)
National Aeronautics and Space Administration
Washington, DC 20546-0001

10. SPONSORING / MONITORING AGENCY REPORT NUMBER
NASA TM-110332

11. SUPPLEMENTARY NOTES
Deng: University of South Carolina, Columbia, SC
Newman: Langley Research Center, Hampton, VA

12a. DISTRIBUTION / AVAILABILITY STATEMENT
Unclassified - Unlimited
Subject Category 24

12b. DISTRIBUTION CODE

13. ABSTRACT (*Maximum 200 words*)

ZIP2DL is a two-dimensional, elastic-plastic finite element program for stress analysis and crack growth simulations, developed for the NASA Langley Research Center. It has many of the salient features of the ZIP2D program. For example, ZIP2DL contains five material models (linearly elastic, elastic-perfectly plastic, power-law hardening, linear hardening, and multi-linear hardening models), and it can simulate mixed-mode crack growth for prescribed crack growth paths under plane stress, plane strain, and mixed state of stress conditions. Further, as an extension of ZIP2D, it also includes a number of new capabilities. The large-deformation kinematics in ZIP2DL will allow it to handle elastic problems with large strains and large rotations, and elastic-plastic problems with small strains and large rotations. Loading conditions in terms of surface traction, concentrated load, and nodal displacement can be applied with a default linear time dependence or they can be programmed according to a user-defined time dependence through a user subroutine. The restart capability of ZIP2DL will make it possible to stop the execution of the program at any time, analyze the results and/or modify execution options, and resume and continue the execution of the program.

This report includes three sections: a theoretical manual section, a user manual section, and an example manual section. In the theoretical section, the mathematics behind the various aspects of the program are concisely outlined. In the user manual section, a line-by-line explanation of the input data is given. In the example manual section, three types of examples are presented to demonstrate the accuracy and illustrate the usage of this program.

14. SUBJECT TERMS
Cracks; Finite-Element method; Elasticity; Plasticity; Fracture; Stress; Strain

15. NUMBER OF PAGES
97

16. PRICE CODE
A05

17. SECURITY CLASSIFICATION OF REPORT	18. SECURITY CLASSIFICATION OF THIS PAGE	19. SECURITY CLASSIFICATION OF ABSTRACT	20. LIMITATION OF ABSTRACT
UNCLASSIFIED	UNCLASSIFIED	UNCLASSIFIED	UNLIMITED